



**Programming-
Manual Milling
V410-V420-V500**

Master book

**HEIDENHAIN NUMERIC B.V.
Eindhoven (NL)
Tel: 31.40.250 13 00
Fax: 31.40.250 13 01**

20/04/2001

© HEIDENHAIN NUMERIC B.V. EINDHOVEN, THE NETHERLANDS 2001

The publisher accepts no liability in respect of specifications on the basis of the information contained in these instructions.

For the specifications of the numerical controller, please refer to the order data and corresponding specification description only.

All rights reserved. Copying of this manual or parts thereof only permitted with the written consent of the copyright holder.

Changes to and errors in this publication cannot be excluded. Therefore no claims can be laid to the information, figures and descriptions contained in this publication.

TABLE OF CONTENTS

TABLE OF CONTENTS.....	1
1. INTRODUCTION.....	1
1.1 Foreword	1
1.2 Companion-manuals	1
1.3 Software version dependency.....	1
2. IMPROVEMENTS OVER ITS PREDECESSORS.....	3
2.1 The V500 improvements	3
2.2 The V420 improvements	3
2.3 The V410.00K improvements	4
2.4 The V410 (till V410.00I) improvements.....	4
3. GENERAL PROGRAMMING INFORMATION.....	5
3.1 About partprograms	5
3.1.1 Program words	6
3.1.2 Program blocks	7
3.1.3 Writing a partprogram	8
3.1.4 Zero points	9
3.2 Axes configurations on machine tools	12
3.2.1 Defining coordinates	14
3.2.2 Cartesian-coordinates	14
3.2.3 Polar coordinates	15
3.2.4 Combining a linear coordinate and angle	16
4. ABOUT THIS MANUAL.....	17
4.1 Philosophy and purpose of the manual.....	17
4.2 Contents of each section	17
4.2.1 G-functions	17
4.2.2 F-functions	17
4.2.3 H-functions	17
4.2.4 M-functions.....	18
4.2.5 S-function	18
4.2.6 T-function	18
4.2.7 E-parameter	18
4.2.8 Geometric calculations with continuous movements	18
4.2.9 Graphical support.....	18
4.2.10 Machine constants	18
4.2.11 External program selection	18
4.3 Programming functions	18
5. G-FUNCTIONS	21
5.1 G0 Rapid traverse	21

TABLE OF CONTENTS

5.2	G1 Linear interpolation	24
5.3	G2/G3 Circular interpolation (CW/CCW)	35
5.4	G4 Dwell time	51
5.5	G6 Spline-interpolation	52
5.6	G7 Tilting working plane	61
5.7	G8 Tilting tool orientation	69
5.8	G9 Define pole position (size reference point)	74
5.9	G11 One/Two point or two line geometry with chamfer or rounding	81
5.10	G14 Repeat function	95
5.11	G17 Mainplane XY, tool Z	98
5.12	G18 Mainplane XZ, tool Y	101
5.13	G19 Mainplane YZ, tool X	104
5.14	G22 Macro call	107
5.15	G23 Main program call	113
5.16	G25/G26 Enable/Disable feed- and/or speed-override	115
5.17	G27/G28 Positioning functions	116
5.18	G29 Conditional jump	120
5.19	G36/G37 Activate/ Deactivate turning mode	123
5.20	G39 Activate/Deactivate tool offset	124
5.21	G40 Cancel tool radius compensation	128
5.22	G41/G42 Tool radius compensation (left/right)	132
5.23	G43/G44 Tool radius compensation to/past endpoint	143
5.24	G45 Axis parallel measuring movement and measuring tool dimensions	147
5.24.1	G45 Measuring a point	147
5.24.2	G45 + M25 Measure tool dimensions	153
5.25	G46 Measuring a full circle or probe calibration	156
5.25.1	G46 Measuring a full circle	156
5.25.2	G46 + M26 Probe calibration	161
5.26	G49 Checking on tolerances	164
5.27	G50 Processing measuring results	169
5.28	G51/G52 Cancel/activate pallet zero point shift	181
5.29	G53/G54—G59 Cancel/activate zero point shift	182
5.29.1	G53/G54—G59 Cancel/activate zero point shift (MC84=0)	182
5.29.2	G54 Extended zero point shift (MC84>0)	186
5.30	G61 Tangential approach	191
5.31	G62 Tangential exit	196
5.32	G63/G64 Cancel/Activate geometric calculations	201
5.32.1	Intersection point between two straight lines	203
5.32.2	Intersection point between line and circle or circle and line	206
5.32.3	Intersection point between two circles	207
5.32.4	Two tangent geometry elements	209
5.32.4.1	Tangency indicator	209
5.32.4.2	A connecting circle between line and circle or circle and line	211
5.32.4.3	A connecting circle between two circles outside each other	213
5.32.4.4	A connecting circle between two circles of which one circle inside the other one	214
5.32.4.5	A connecting circle between two concentric circles	215
5.32.5	Line definitions	216
5.32.5.1	Parallel line	218
5.32.5.2	Intersection point indicator	219
5.33	G66/G67 Select negative/positive tool direction	224
5.34	G70/G71 Inch/Metric programming	227
5.35	G72/G73 Cancel/Activate scaling or mirror imaging	229
5.36	G74 Absolute position	237
5.37	G77 Bolt hole cycle	241
5.38	G78 Point definition	246
5.39	G79 Activate cycle	250
5.40	G81 Drilling cycle	254
5.41	G83 Deep hole drilling cycle	257
5.42	G84 Tapping cycle	262
5.43	G85 Reaming cycle	266

5.44	G86 Boring cycle	269
5.45	G87 Rectangular pocket milling	272
5.46	G88 Groove milling cycle	278
5.47	G89 Circular pocket milling cycle	284
5.48	G90/G91 Absolute/incremental programming	289
5.48.1	G90/G91 Absolute/incremental programming	289
5.48.2	Wordwise absolute and incremental programming	292
5.49	G92/G93 Incremental/Absolute zero point shift	294
5.50	G94/G95 Select feedrate unit	302
5.51	G96/G97 Constant cutting speed	304
5.52	G98 Graphic window definition	305
5.53	G99 Definition of workpiece blank as a box	307
5.54	G141 3D-Tool correction with dynamic TCMP	309
5.55	G145 Linear measuring movement	321
5.56	G148 Reading measure probe status	340
5.57	G149 Reading tool data or zero offset values	342
5.58	G150 Change tool data or zero offset values	346
5.59	G174 Tool withdrawal movement	349
5.60	G180 Basic coordinate system	351
5.61	G182 Cylindrical coordinate system	353
5.62	G195 Graphic window definition	362
5.63	G196 End contour description	364
5.64	G197/G198 Begin inner/outer contour description	365
5.65	G199 Begin contour description	373
5.66	G200---G208 Pocket Cycle	381
5.66.1	G200 Begin pocket cycle	381
5.66.1.1	Introduction universal pocket cycle	381
5.66.1.2	Part program structure	383
5.66.1.3	Translation, rotation and mirror image of a pocket	384
5.66.1.4	Same pocket in another program	385
5.66.1.5	Operating section	386
5.66.1.6	Error messages	388
5.66.2	G201 Start contour pocket cycle	394
5.66.2.1	Usage of the generated macros	396
5.66.2.2	Macro for finishing a pocket contour	398
5.66.2.3	Sequence of the macros on the machine	401
5.66.3	G202 End contour pocket cycle	406
5.66.4	G203 Start pocket contour description	409
5.66.5	G204 End pocket contour description	412
5.66.6	G205 Start island contour description	414
5.66.6.1	Description of the island contour	415
5.66.7	G206 End pocket contour description	419
5.66.8	G207 Call island contour macro	422
5.66.9	G208 Quadrangle contour description	425
6.	SPECIFIC G-FUNCTIONS FOR MACROS	431
6.1	Overview:	431
6.2	Error message functions	432
6.2.1	G300 Programming error messages	432
6.2.2	G301 Error in program or macro that just has been read in.	434
6.3	Executable functions	435
6.3.1	G302 Overwriting radius compensation parameters	435
6.3.2	G303 M19 with programmable direction	436
6.4	Query functions	437
6.4.1	G319 Query actual technology data	437
6.4.2	G320 Query current G data	439
6.4.3	G321 Query tool data	443
6.4.4	G322 Query machine constant memory	445

TABLE OF CONTENTS

6.4.5	G324 Query G-group	446
6.4.6	G325 Query M group	448
6.4.7	G326 Query actual position	450
6.4.8	G327 Query operation mode	452
6.4.9	G329 Query programmable kinematic model	453
6.5	Write functions	454
6.5.1	G331 Write tool data	454
6.5.2	G339 Write programmable kinematic model	457
6.6	Formatted write functions	459
6.6.1	File to define an array and fill it with basic settings	459
6.6.2	E-Parameter Array	461
6.6.3	Configuration file to define a file or window (display/input)	462
6.6.4	G350 Writing to a window	463
6.6.5	G351 Writing to a file	466
7.	TOOL MEASURING CYCLES FOR LASER MEASURING	469
7.1	General remarks	469
7.2	G600 Laser: Calibration	470
7.3	G601 Laser: Measure tool length	473
7.4	G602 Laser: Measure length and radius	475
7.5	G603 Laser: Check of individual edge	477
7.6	G604 Laser: Tool breakage control	479
8.	MEASURING SYSTEM TT130	481
8.1	General notes	481
8.2	G606 Calibration TT130	482
8.3	G607 Measuring tool length TT130	484
8.4	G608 Measuring tool radius TT130	488
8.5	G609 Measuring length and radius TT130	490
9.	CYCLE DESIGN	493
9.1	Introduction	493
9.2	Description of G function and addresses (G5?? CFG)	495
9.2.1	Example- G5?? .CFG file (definition G5?? .CFG	495
9.2.2	Example-G550.CFG file	497
9.2.3	Permitted addresses	498
9.3	Support graphics	499
9.3.1	Making graphics in *.DXF format	499
9.3.2	Making graphics in *.BMP format	499
9.3.3	Colour code	499
9.3.4	Types of line	500
9.4	Execution macro	501
9.4.1	Explanation	501
9.5	Reading cycle files into the CNC	502
10.	TECHNOLOGICAL INSTRUCTIONS	503
10.1	F-Function	503
10.2	H Auxiliary function	508
10.3	M-Funktionen	509
10.3.1	M0 Program stop	509
10.3.2	M1 Optional program stop	510

10.3.3	M3/M4 Spindle-rotating clockwise/counter clockwise	511
10.3.4	M5 Spindle stop	512
10.3.5	M6 Automatic tool change	513
10.3.6	M7/M8 Switch on coolant supply nr 2 / nr. 1	515
10.3.7	M9 Switch off coolant supply	516
10.3.8	M13/M14 Switch on nr. 1 coolant and rotate spindle CW/CCW	517
10.3.9	M19 Oriented spindle stop	519
10.3.10	M25 Measuring tool sizes	520
10.3.11	M26 Calibration the measuring probe	521
10.3.12	M24/M27/M28 Switch on/off a measuring probe	522
10.3.13	M30 End of partprogram	523
10.3.14	M41/M42/M43/M44 Select spindle speed range	524
10.3.15	M66 Manuel tool change	526
10.3.16	M67 Change tool compensation values	528
10.4	S-function	529
10.5	T-function tool number and tool memory	530
10.5.1	T-function	530
10.5.2	Tool memory	534
11.	E-PARAMETERS AND ARITHMETIC FUNCTIONS	541
11.1	E-Parameter	541
11.2	Arithmetical functions	543
12.	TURNING MODE	553
12.1	Introduction	553
12.2	Machine constants	554
12.3	G36/G37 Activate/deactivate turning mode	555
12.4	G17 Mainplane for turning mode (G17 Y1=1 Z1=2)	556
12.5	G94/G95 Expanded choice of feed unit	557
12.6	G96/G97 Constant cutting speed	558
12.7	Defining turning tools in the tool table	559
12.8	G302 Overwriting radius compensation parameters	560
12.9	Unbalance cycles	562
12.9.1	General information	562
12.9.2	Description of unbalance	562
12.9.3	G691 Unbalance measurement	563
12.9.4	G692 Unbalance check	566
12.9.5	Unbalance example	567
12.10	Turning cycles	568
12.10.1	G822 Clearance axial	569
12.10.2	G823 Clearance radial	571
12.10.3	G826 Clearance axial finishing	573
12.10.4	G827 Clearance radial finishing	575
12.10.5	G832 Roughing axial	577
12.10.6	G833 Roughing radial	579
12.10.7	G836 Roughing axial finishing	581
12.10.8	G837 Roughing radial finishing	583
12.10.9	G842 Grooving axial	585
12.10.10	G843 Grooving radial	587
12.10.11	G846 Grooving axial finishing	589
12.10.12	G847 Grooving radial finishing	591
12.11	Examples	593
13.	G64 GEOMETRIC CALCULATIONS WITH CONTINUOUS MOVEMENTS	597

TABLE OF CONTENTS

13.1	Intersection point	599
13.1.1	Intersection point of two straight lines	599
13.1.2	Intersection point programmed as end point	602
13.1.3	Chamfer between intersecting straight lines	605
13.1.4	Rounding between intersecting straight lines	606
13.1.5	Rounding between straight line and chamfer	607
13.1.6	Intersecting point between line circle	608
13.1.7	Intersecting point of line and circle programmed as end point	610
13.1.8	Rounding between intersecting line and circle	612
13.1.9	Intersecting point between circle and line	614
13.1.10	Intersecting point of circle and line programmed as end point	616
13.1.11	Rounding between intersecting circle and line	618
13.1.12	Intersecting point between two circles	620
13.1.13	Intersection point between two circles programmed as end point	622
13.1.14	Rounding between two intersecting circles	624
13.2	Point of tangency	625
13.2.1	Point of tangency indicator (R1=)	625
13.2.2	Tangent line and circle	625
13.2.3	Continuous connecting circle between tangent line and circle	629
13.2.4	Tangent circle and line	630
13.2.5	Continuous connection circle between tangent circle and line	633
13.2.6	Tangent circle and line	634
13.2.7	Continuous connecting circle between two tangent circles	635
13.3	Continuous connecting circle between elements which do not meet	636
13.3.1	Line and circle	636
13.3.2	Circle and line	638
13.3.3	Two circles outside each other	640
13.3.4	One circle inside the other one	641
13.3.5	Concentric circles	642
13.4	G64 Geometric calculations with non-continuous movements	643
13.4.1	Rounding or connecting circle indicator (K1=)	643
13.4.2	Rounding with intersection points	643
13.4.3	Rounding between intersecting straight lines	644
13.4.4	Rounding between intersecting line and circle	646
13.4.5	Rounding between intersecting circle and line	647
13.4.6	Rounding between two intersecting circles	648
13.4.7	Tangent lines (R1=)	649
13.4.8	Connecting circle between a line tangent to a circle or v.v.	650
13.4.9	Connecting circle between a line which does not meet a circle	652
13.4.10	Connecting circle between circles outside each other	654
13.4.11	Connecting circle between two circles one inside the other	656
13.4.12	Connecting circle with two concentric circles	658
13.5	Examples	659
14.	APPENDIX	679
14.1	Tilting of the operating plane	679
14.1.1	Introduction	679
14.1.2	Machine types	680
14.1.3	Kinematics model	682
14.1.4	Operations	683
14.1.4.1	Manual operations	683
14.1.4.2	Display	683
14.1.4.3	Axis selection/position axis	684
14.1.4.4	Reference point	684
14.1.4.5	Intervention	684
14.1.5	Error messages	684
14.1.6	Machine Constants	685
14.2	Look Ahead Feed (LAF) function	686

14.2.1	Introduction	686
14.2.2	Detailed specification	686
14.3	Graphical support.....	688
14.3.1	Synchron graphic	688
14.3.2	Graphical simulation	688
14.3.3	Graphical programming support	690
14.4	Machine constants	692
14.5	External program selection	695
INDEX.....		697

1. Introduction

1.1 Foreword

This manual assists you in programming the controller.

The machine should not be operated, even for a short period, by anyone who has not received the necessary training either in the Company, at an Institute of Further Education or in one of the Training Centres.

Please follow this advice to ensure proper usage.

The controller and the machine are coordinated using machine constants. Some of these constants are accessible to the user. Caution!

A thorough understanding of the significance and functions of these constants is required if they are to be changed. If in doubt please contact our Customer Service Department.

Users should therefore always output their programs and specific data (e.g. technical data, machine constants etc.) on their PC or onto diskette. This prevents data from being lost irretrievably if the battery or back-up battery is defective.

We reserve the right to change the design, equipment and accessories in the interest of further development. No liability will be accepted for any errors in the data, illustrations or descriptions.

1.2 Companion-manuals

The information relating to the installation, interfacing, operation, and programming for the controller cannot be adequately described in a single manual. Therefore, several manuals have been designed to give the user information relating to a particular type of task. The set of manuals available for the controller is listed in this section.

- User Manual
- CDS Manual (CNC Data Station Manual)
- Installation Manuals
- Interfacing
 - MIPS (Machine Interface Programming System.)
 - Basic IPLC Program

1.3 Software version dependency

If the control unit has a V410 version, the functionality improvements of the V420\V500, such as described in chapter 2, are not available.

2. Improvements over its predecessors

2.1 The V500 improvements

The Versions V420 and V500 are for the programming Identical. Only the Hardware is changed.

2.2 The V420 improvements

Added functions:

- Dynamic TCPM (Tool Centre Point Management) **(G141)**
- Tool withdrawal movement **(G174)**
- M19 with programmable direction **(G303)**
- Query current G data **(G320)**
- Query programmable kinematic model **(G329)**
- Write programmable kinematic model **(G339)**
- Writing to a window **(G350)**
- Writing to file **(G351)**

- Turning Chapter

G17 extended	
G36 and G37	Turning mode
G96 and G97	Constant Cutting speed
G302	Override radius compensation parameters
G691	Unbalance measurement
G692	Unbalance check
G822, G823, G826 und G827	Clearance
G832, G833, G836 und G837	Roughing
G842, G843, G846 und G847	Grooving

Modified functions:

- Begin graphic model description **(G198)** extended with I1= address.
- Begin outside contour description **(G199)** extended with B=4.
- Read actual technology data **(G319)** extended with S1 address.
- Read tool data **(G321)** extended with O address.
- Read G-group **(G324)** extended.
- Write tool data **(G331)** extended with O address.
- The number of possible addresses in Cycle Design is expanded and OPTIONAL and ACTIVE have been added.
- Calculation for E-parameters extended with the floor(E2), ceil(E2), round(E2,n), mod(E2,E3), sign(E2), atan(E2,E3), acos(E2,E3) and asin(E2,E3) functions.

2.3 The V410.00K improvements

Added functions:

- G66** and **G67** T-length compensation in negative or positive direction

Modified functions:

- Tilting tool orientation (**G8**) extended with address L3=
- Acceleration reduction (I6=.. in **G27/G28**)
- Lasermesszyklen (**G600-G604**) extended with addresses.

2.4 The V410 (till V410.00I) improvements

Added functions:

- G8** Tilting tool orientation
- G327** Scan active operating mode
- G600-G609** Tool measurement cycles for laser measurement and TT130.
- Cycle Design**

Modified functions:

- Calculated rotary axis radius (**G0/G1**).
- Programmable contour accuracy (**G28**).
- Tapping with active G7 in main plane also with open control circuit (**G84**)
- Reset function (**G93**)
- Activating rotary axis calculation (**G94**).
- Modal G function (**G324**) is extended by I1=28.

3. General programming information

3.1 About partprograms

A partprogram is the complete set of data and instructions required for producing a particular workpiece on a numerically controlled machine tool.

The instructions may contain different operations, such as milling, drilling, tapping, etc. Each separate operation is a unit, which can be split up into smaller instructions. One program block specifies one complete operation. The words in a block define the smaller instructions.

The proper machining sequence, with all the separate instructions, must be stated in a partprogram. Examples of separate instructions are tool movements, machine tool functions and technological data.

A program cannot be executed until it has been properly stored in the CNC system memory. A partprogram can be created and stored into the CNC memory in different ways:

1. Use interactive contour programming (ICP) for complex contours.
2. Use interactive partprogramming (IPP) for programming without knowledge of DIN programming.
3. Enter the program manually via the control panel.
4. Create the program separate from the control, use data terminal equipment to produce a data carrier (such as a punched paper tape, a magnetic digital cassette or disk) and input the data into the CNC memory.

3.1.1 Program words

The CNC PILOT control system employs the standard WORD ADDRESS system in which a word has two parts:

1. -The addresses, which can be a single address (one alpha character) or an indexed address. An indexed address has an alpha character followed by an index and the character =, e.g. E1=.
2. A multi-digit number.

Words do not need leading zeros. However, if the value of a word is zero, then at least one zero must be written.

The words stating dimensional information can have a plus or a minus sign. If no sign is programmed, a positive value is assumed. A negative value must have a minus sign. Dimension words can be written with a decimal point; therefore trailing zeros need not be stated. The control system assumes that the decimal point is behind the last digit of the number if the decimal point is not stated.

MODAL WORDS

A modal word stays active after the execution, until the word is used again or reset.

NON-MODAL WORDS

A non-modal word is only active in the block. When necessary it must program again.

Example of a single and an indexed word

Single word: X-21.43

'X' is the address, '-' is the sign and '21.43' the decimal number.

Indexed word: X1=-21.43

'X1=' is the address, '-' is the sign and '21.43' the decimal number.

3.1.2 Program blocks

A block can include several words considered as a unit, which contains all the information needed for one complete operation or function. This operation can be a tool movement or a machine tool function, or a combination of both.

The CNC PILOT control system employs a VARIABLE BLOCK FORMAT. The block lengths can be different because of changes in the number or length of the words. A block can contain up to 255 characters.

The N-word must always be the first word in a block. The other words can be written in any order. The example gives the preferred sequence for the frequently used words.

Each word can occur only once in a block. Words such as E1= and E2= have different addresses and therefore can be both present in the same block.

On a data carrier the character line feed [LF] separates the blocks.

Example of a program block

```
N20 G1 X14 Z62.5 F300 S200 T12 M3
```

N20: Block number

G1: Preparatory function

X14 and Z62.5: Dimensional information

Technological and machine data such as spindle speed (S), feed rate (F), tool selection (T) and E.g. a direction of spindle rotation (M3) may be included as well.

THE BLOCK NUMBER N

The first word in a block is the block number, which identifies that block. Each block must have a separate number. The block numbers range from N0 to N9999999.

A general rule is that a block number cannot be in the same program more than once. However, the check on the block numbers is inactive if a machine constant is set or the BTR possibility is used. The machine constant setting is useful when large programs should be executed and the BTR possibility is not used.

Block numbers can be in any sequence. The execution will be in the programmed sequence e.g.

Programmed sequence: N10, N50, N30

Executed sequence: N10, N50, N30

The re-number function of the control allows the block numbers to be automatically renumbered in increasing order, starting from N1.

The CNC system automatically generates block numbers when the programmer uses the control panel to input programs.

3.1.3 Writing a partprogram

PROGRAM IDENTIFICATION

Each partprogram or subprogram has to start with an identification number, which ranges from 1 to 9999999.

So numbers as 1, 125, 9001, 12345, 876543, 3451592 are valid identification numbers.

The rename function of the control is available for changing the identification number.

A partprogram name can be written between the characters CONTROL OUT '(' and CONTROL IN ')' and immediately after the identification number. These names are listed if the file directory is displayed on the control.

Example of partprogram identification with name
9001 (PLATE NR. A334)

The following partprogram identification is possible for earlier CNC systems (compatibility) (for programs %PM... and macros %MM...)

%PM9001
N9001 (PLATE NR. A334)

These programs are automatically identified and stored correctly by the CNC system. Data transmission from the CNC to the outside is controlled by machine constants (MC795, MC799)

PARTPROGRAM SETUP

To write a partprogram the programmer must do the following:

1. Determine the mounting of the workpiece and the position of the clamps
2. Determine the machining operation sequence
3. Determine the tools required for the operations
4. Determine for each tool the appropriate technological data
5. Determine the workpiece dimensions and the necessary movements.

The points 1 to 4 are outside the scope of this manual.

The movements on the machine are a combination of tool and workpiece movements. To simplify the programming the programmer should assume that all movements are tool movements. The configuration of the machine tool and CNC system determines how the movements are actually performed.

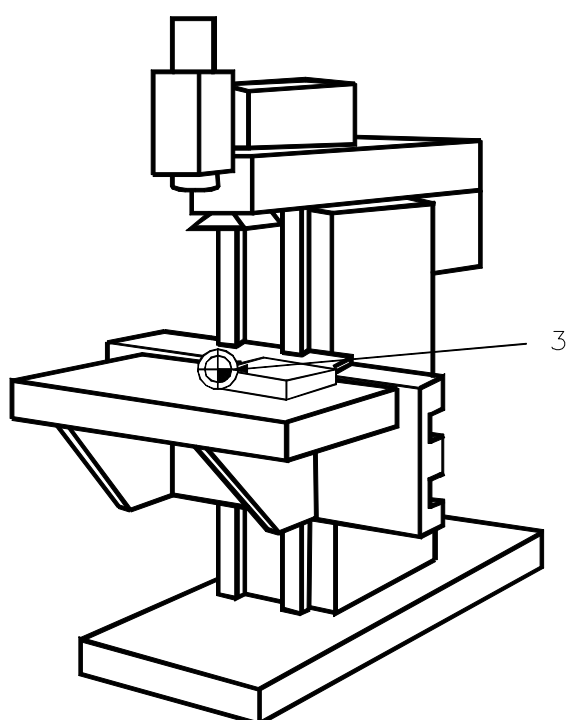
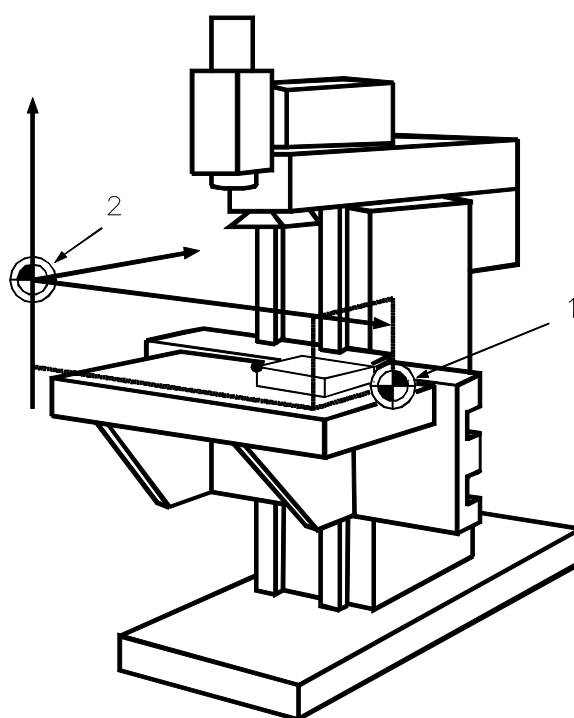
An imaginary coordinate system is positioned on the workpiece so that programmed movements refer to a zero. The programmer determines the position of this point in such a manner that the easiest programming calculations are produced. Refer to Axes configurations on machine tools for the directions of the coordinate axes.

PROGRAM STORAGE

The user memory can store more than one partprogram or subprogram (macro). The actual number of stored programs and macros depends upon the size of each program or macro and on the available memory capacity. A machine constant sets the maximum amount of programs to be between 16 and 1000.

With the lock function it is possible to protect partprograms and macros against unauthorised editing on the control.

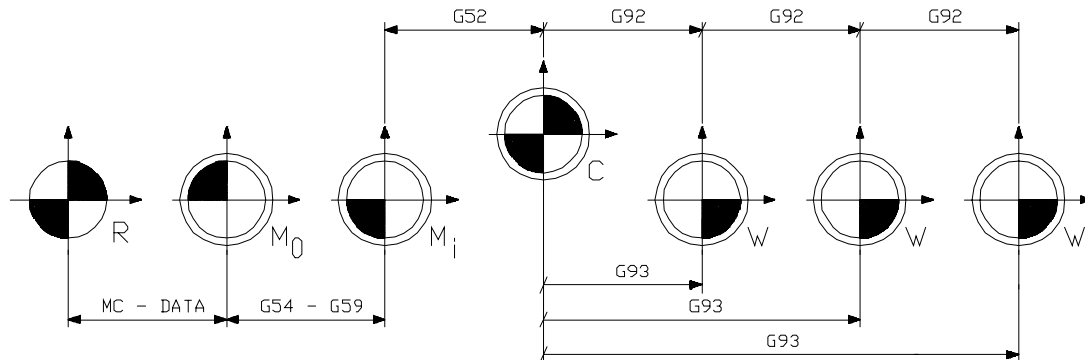
3.1.4 Zero points



- 1 = Machine reference point
- 2 = Machine zero point
- 3 = Program zero point

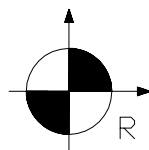
After power-on, REFERENCE POINT SEARCH must be carried out first. As a result, the machine zero point is known, since the zero offsets from the machine zero point (M0) to the machine reference point (R) are stored as machine constants.

The partprogrammer establishes a program zero point (W), which is related to the part and from which the part dimensions are measured. This program zero point must also be related to the machine zero point, which can be established with the functions G52 and/or G54-G59.



1. Machine reference point (R)

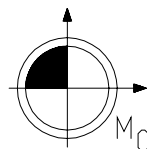
Each axis of a machine tool has a fixed point called the reference point of the axis. The reference points of all axes form the machine reference point (R).



During REFERENCE POINT SEARCH (refer to the Operating Manual) the tool moves to the reference point of the selected axis (or axes). When the reference point is reached, the axis is automatically zeroed by the control and the positions of the software limit switches are set.

2. Machine zero point (M0)

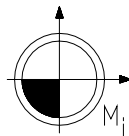
The machine zero point is also a fixed point on the machine.



When the CNC control system is commissioned, the distances from the machine reference point (R) to the machine zero point (M0) are measured along the axes and stored in the machine constant memory. Each axis has its own machine constant for this purpose.

After the machine reference point is established by REFERENCE POINT SEARCH, the control system reads the associated dimensions from the machine constant memory. The machine zero point (M0) is set as the origin of the coordinate system and the displayed positions are related to this zero point.

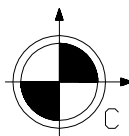
3. Secondary machine zero point (Mi)



When the machine tool has several clamping stations (e.g. pallet stations) each clamping station must have its own fixed zero point. These fixed zero points are called secondary machine zero points (Mi)

The zero offset memory contains the axis distances between the machine zero point (M0) and the secondary machine zero points (Mi). Six secondary points can be stored by using the G54 to G59, G54I[0..99](from V320) functions.

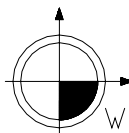
4. Mounting zero point (C)



When a secondary machine zero point (Mi) is established, the zero point of the mounting device must be determined. This zero point may coincide with the active Mi or can be set by the G52 PRESET AXIS function.

Zero point C is automatically set by the control, when an external program call with offset values is made

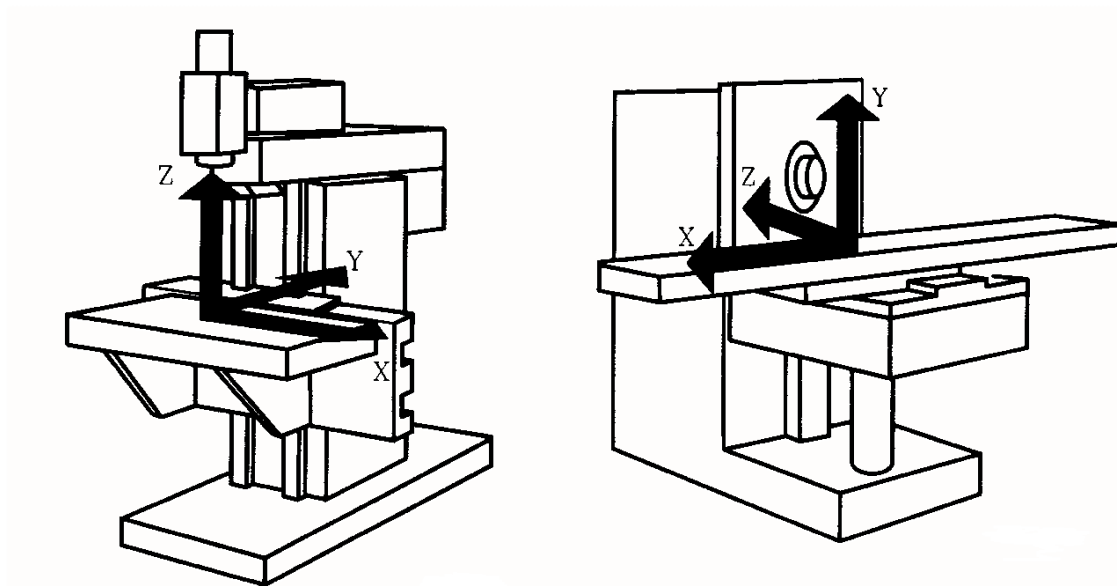
5. Program zero point (W)



The program zero point W is the zero point from which the axis coordinates in the partprogram are measured. The programmer can set the position of point W arbitrarily. The position should be chosen in order that additional calculations for programming the workpiece are kept to the minimum.

The functions G52, G54-G59, G54I[0..99] and G92/G93 establish the relation between the program zero points and the machine zero point.

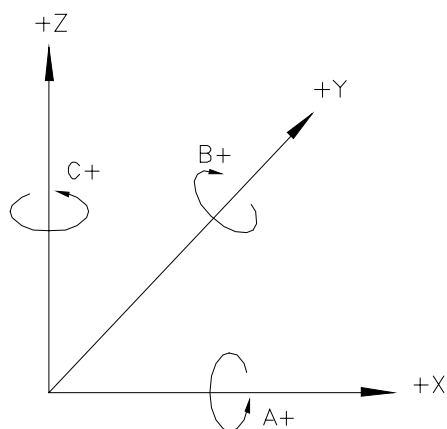
3.2 Axes configurations on machine tools



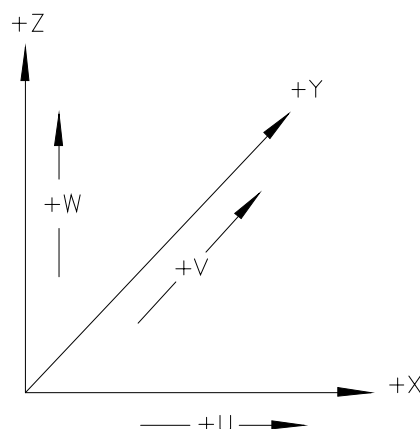
Vertical knee milling machine

Horizontal knee milling machine

A milling machine has three main linear axes (X, Y, Z), which are at 90° to each other. The orientation of these axes is established by the Z-axis, which is always parallel to the main spindle of the machine tool. The X-axis is horizontal and parallel to the work holding surface. Each main axis can have a rotary axis and a linear axis, parallel to a main axis. These are shown in the illustration below.



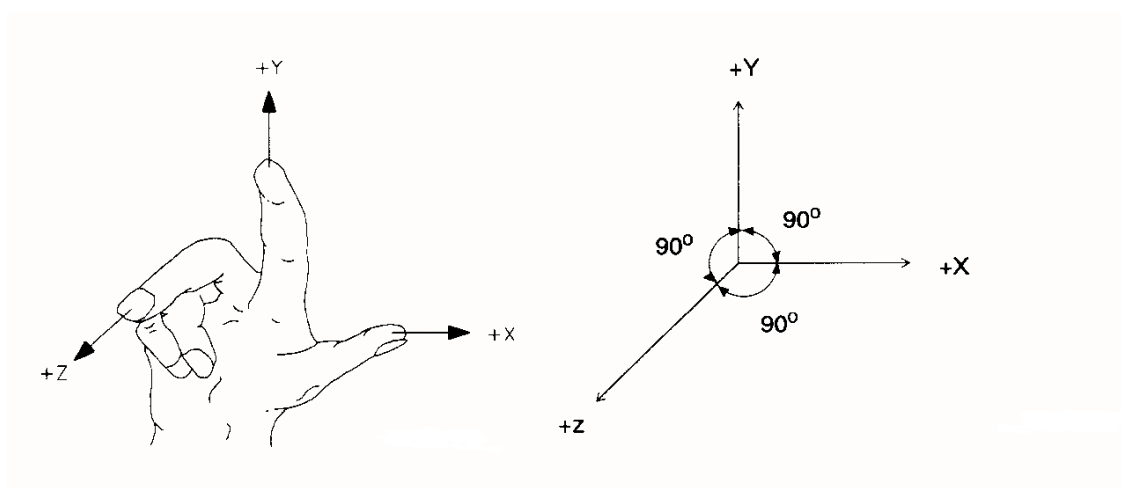
Rotary axes



Linear axes parallel to main axes

Orientation of main axes, rotary axes and linear parallel axes.

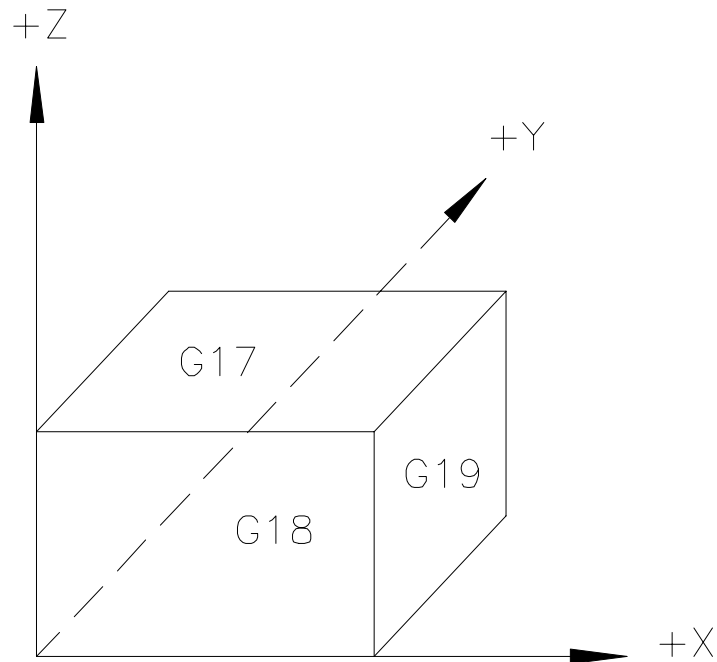
The standards ISO 841, DIN 66217 and EIA RS-267-A, all define the positions of axes on a numerically controlled machine. The right-hand rule is used for stating the orientation for all CNC machines axes.



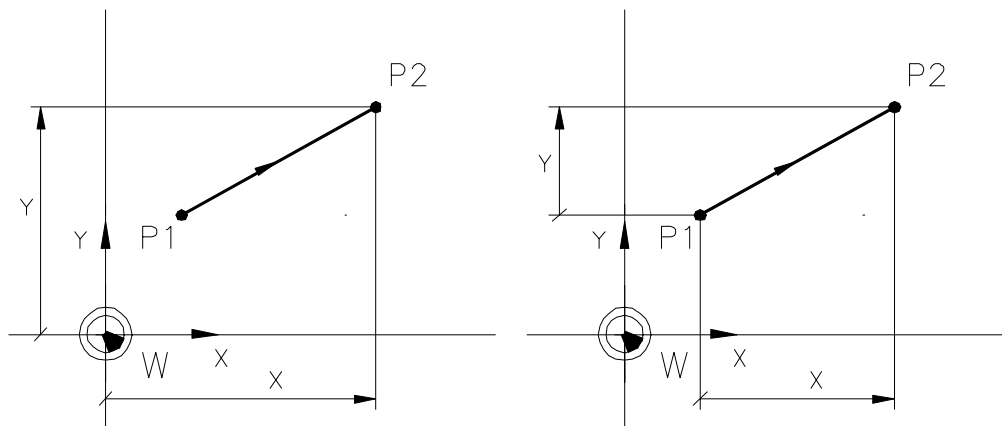
The thumb represents the X-axis, the forefinger the Y-axis and the middle finger the Z-axis. The directions in which the fingers are pointing represent the positive directions along the axes.

3.2.1 Defining coordinates

Axes distances define the coordinates of points in three-dimensional (3-D) space. Axis coordinates will be in one of three planes (XY-plane, XZ-plane, YZ-plane).



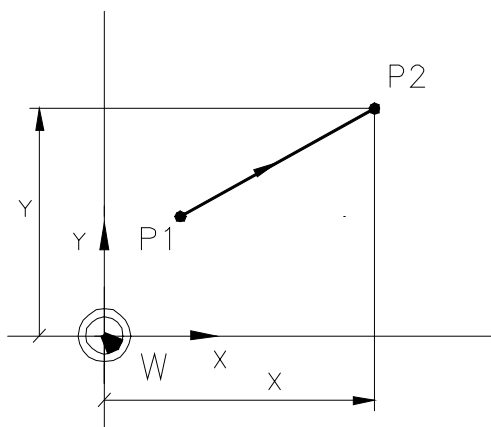
3.2.2 Cartesian-coordinates



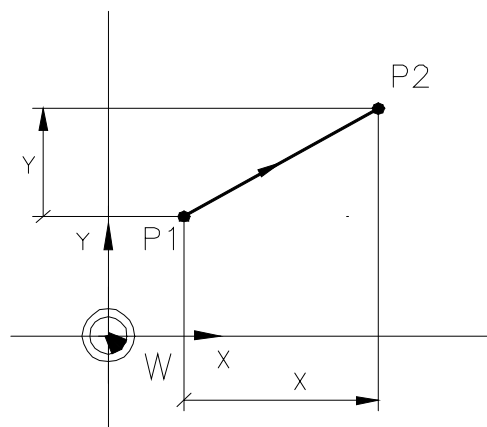
Absolute (G90) coordinates

Incremental (G91) coordinates

3.2.3 Polar coordinates

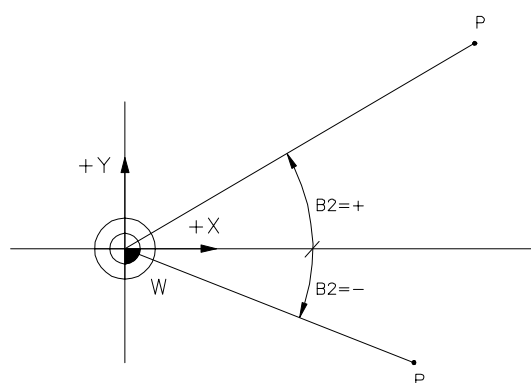


Absolute (G90) coordinates

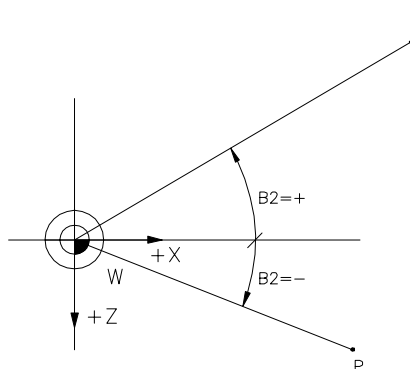


Incremental (G91) coordinates

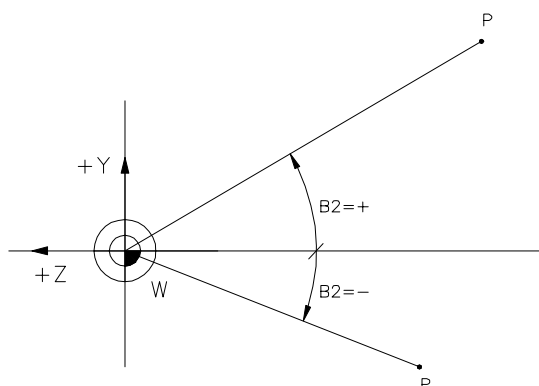
XY-PLANE (G17)



XZ-PLANE (G18)

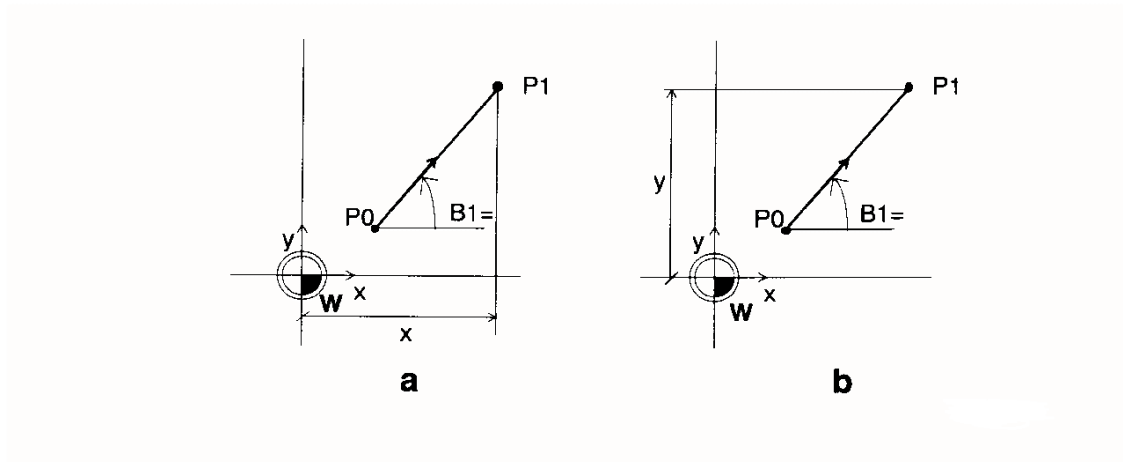


YZ-PLANE (G19)

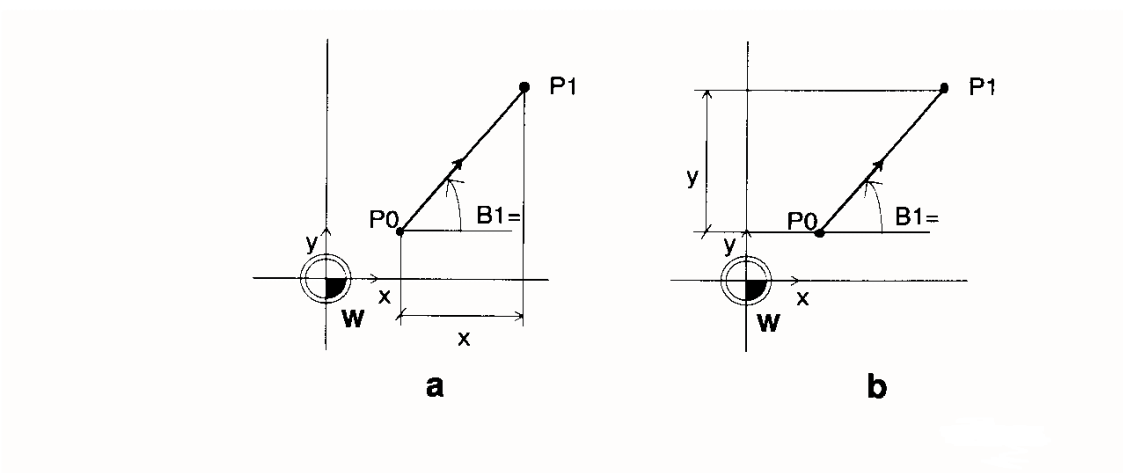


3.2.4 Combining a linear coordinate and angle

One linear axis coordinate and an angle can, in combination, define a point's position.



Absolute (G90) coordinates



Incremental (G91) coordinates

4. About this manual

4.1 Philosophy and purpose of the manual

This manual has been arranged to allow access to comprehensive information relating to programs for the CNC PILOT system.

The core of the reference data in this manual is contained within the sections, which describe the F, G, H, M, S and T- functions. Additional information, such as mathematical operations and formula, is contained in appendices.

4.2 Contents of each section

4.2.1 G-functions

G-functions are functions preparing the CNC controlled machine to programming instructions and are therefore named Preparatory functions. The contents of each section describing a G- function is placed under standard headings, which are:

Purpose

The reason(s) for using the function

Format

The format(s) of a program block containing the function. The conventions used under this heading are given in the Introduction section of this manual.

Parameters

Program words defining the extent of the function's influence, or words, which can be programmed, when the function is active.

Associated functions

Functions, which are in the same functional group, they can cancel the function.

Type of functions

Explains if the function is modal or not.

Notes and usage

Explanation(s) of how the function is used and under what circumstances.

Examples

Examples showing practical applications of the function.

4.2.2 F-functions

F-functions are functions, which establish the feedrate (mm/min or inch/min). The feed rate is dependent of the situation.

4.2.3 H-functions

H-functions are assigned tasks by the machine tool builder. The programmer must therefore read the machine tool builder's documentation for description(s) of how this auxiliary function is used.

4.2.4 M-functions

M-functions are functions, which directly affect CNC machine operations, e.g. switching the coolant supply on and off.

4.2.5 S-function

The S-function specifies the spindle speed in revolutions per minute (RPM).

4.2.6 T-function

The T-function specifies the number, which is used to select a tool and also to store its dimensions in the CNC's Tool Memory.

4.2.7 E-parameter

E-parameter is useful in making a program more flexible. One program can be used for different products.

4.2.8 Geometric calculations with continuous movements

Geometric calculations with continuous movements is a function which able the operator to make a program without knowing the exact coordinates of a certain point.

4.2.9 Graphical support

Graphical support visualizes the movement of the tool in different ways on the display.

4.2.10 Machine constants

This chapter explains how the machine manufacturer can customize the controller for his machines.

4.2.11 External program selection

This chapter explains the use of automatic workpiece changers.

4.3 Programming functions

Fundamentals of CNC Programming

Coordinate Measurement Modes

- G90/G91 - Absolute/incremental programming.
- G70/G71 - Inch/metric programming.

Basic Tool Movements

- G0 - Rapid traverse.
- G1 - Linear interpolation.
 - Tool movement with a feedrate (in a linear and a rotary movement)
 - Tool movement with a feedrate (3D-interpolation)
- G2/G3 - Circular interpolation (CW/CCW).
- G78 - Point definition.

Radius Compensation

- G41/G42 - Tool Radius Compensation (Left/Right).
- G43/G44 - Tool Radius Comp. TO/PAST End Point.
- G40 - Cancel Tool Radius Compensation.

Main Planes

- G17 - Main plane XY, tool Z.
- G18 - Main plane XZ, tool Y
- G19 - Main plane YZ, tool X
- G7 - Tilt operating planes

Positioning & Feedrate Functions

- G27/G28 - Cancel/activate positioning function.
- G25/G26 - Feed override active/inactive.
- G94/G95 - Select feed rate unit.
- G4 - Dwell time.

Tool & Spindle Speed Functions

- S - Spindle speed.
- T - Tool number.

Zero Datum Points

- G51/G52 - Cancel/activate G52 zero point shift.
- G53/G54-G59 - Cancel/activate zero point shift
- G54 I[0..99] - Activate zero point shift
- G92/G93 - Incremental/absolute zero point shift.

Graphical Simulations

- G195 - Graphic window definition.
- G196 - End contour description.
- G197 - Begin inner contour description.
- G198 - Begin outer contour description
- G199 - Begin contour description.

Machine Functions

- M3/M4 - Spindle clockwise/counter-clockwise.
- M5 - Spindle stop
- M19 - Orientated spindle stop
- M7 - Switch on number 2 coolant supply.
- M8 - Switch on number 1 coolant supply.
- M9 - Switch off coolant supply.
- M13 - Switch on No. 1 coolant rotate spindle clockwise.
- M14 - Switch on No. 1 coolant rotate spindle counter-clockwise.
- M6 - Automatic tool change.
- M66 - Manual tool change
- M67 - Change tool compensation values
- M0 - Program stop.
- M1 - Optional program stop.
- M30 - Partprogram end.

Geometric Functions

- G11 - Polar coordinate, corner rounding, chamfer.
- G63/G64 - Cancel/activate geometric calculations.
- G72/G73 - Cancel/activate scaling or mirror imaging.
- G9 - Polar point definition

Defined (Canned) Cycles

- G79 - Activate cycle.
- G77 - Bolthole circle.
- G81 - Drilling cycle.
- G83 - Deep hole drilling cycle.
- G84 - Tapping cycle.
- G85 - Reaming cycle.
- G86 - Boring cycle.
- G87 - Rectangular pocket milling cycle.
- G88 - Slot milling cycle.
- G89 - Circular pocket milling cycle.

Transfer of Program Control

- G14 - Repeat function.
- G29 - Conditional jump.
- G22 - Macro call.
- G23 - Program call.

E-parameters and arithmetical operations

Special Functions

- G141 - 3D tool correction.
- G180/G182 - Cancel/activate cylinder interpolation.
- G6 - Spline interpolation.
- G8 - Swivel tool
- G39 - Activate/deactivate compensation

Measuring Cycles

- G45 - Axis parallel measuring movement.
- G45 M25 - Measure tool dimensions.
- G46 - Measuring a full circle.
- G46 M26 - Probe calibration.
- G49 - Checking on tolerances.
- G50 - Processing measuring results.
- G145 - Linear measuring movement.
- G148 - Read probe status.
- G149 - Read tool data and offsets.
- G150 - Write tool data and offsets.

Auxiliary Function

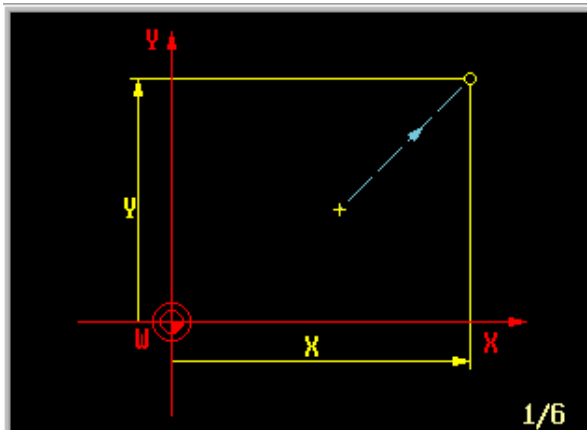
H

5. G-Functions

5.1 G0 Rapid traverse

Purpose

To use the rapid traverse rate for axis movements. This traverse rate is set by Machine Constants (per axis). The function G0 is used mainly for positioning a tool before and after cutting passes.



Format

N... G0 [axis coordinates]

Parameter

```
G    Rapid traverse
X    Endpoint coordinate
Y    Endpoint coordinate
Z    Endpoint coordinate
A    Endpoint angle
B    Endpoint angle
B1=  Angle
B2=  Polar angle
?90= Endpoint abs. (X,Y,Z..)
?91= Endpoint incr. (X,Y,Z..)
L1=  Path length
L2=  Polar length
P1=  Point definition number
```

Modal Words

F, S, T, M, H, E..=

F1=, T1=, T2=

D (Angle oriented spindle stop)

Type of function

Modal

Associated functions

G1, G2/G3, G6, G74

Wordwise absolute/incremental programming (X90=..., X91=..)

Notes and usage**DEFAULT MODE**

The G0 function is automatically set at the start of a program or after CLEAR CONTROL.

CANCELLATION

The function G0 is cancelled by a G1, G2, G3, and G6.

STOP AFTER A RAPID MOVEMENT

The programmed position is reached before the next movement starts. So a stop occurs after a rapid movement.

NO STOP AFTER A RAPID MOVEMENT

If required, rapid movements can be executed without a stop. The function G28 and parameter I4= are used to state, that rapid movements are executed with (default) or without a stop.

Refer to the G28 function RAPID TRAVERSE MOVEMENTS for additional Information about G28 and I4=.

MOVEMENTS IN THE MAIN PLANE

Rapid movements in the main plane, thus the plane defined with G17, G18 or G19, are executed under full control of the linear interpolator. So a straight line is made.

POLAR COORDINATES OR ONE COORDINATE AND ANGLE

Positions in the main plane can also be programmed with polar coordinates or one coordinate and angle.

POSITIONING LOGIC

The positioning logic is a fixed sequence of axis movements depending on the active main plane and the movements along the tool axis.

When the tool is moving towards the workpiece:

	G17 XY-PLANE	G18 XZ-PLANE	G19 YZ-PLANE
1st movement	4th axis	4th axis	4th axis
2nd movement	X and Y	X and Z	Y and Z
3rd movement	Z-axis	Y-axis	X-axis

When the tool is moving away from the workpiece:

	G17 XY-PLANE	G18 XZ-PLANE	G19 YZ-PLANE
1st movement	Z-axis	Y-axis	X-axis
2nd movement	X and Y	X and Z	Y and Z
3rd movement	4th axis	4th axis	4th axis

5TH and 6TH AXES

If these axes are programmed, movements along them will occur simultaneously with the 4th axis.

TOOL IN POSITIVE DIRECTION OF TOOL AXIS (G66/G67)

With G67 the tool is pointing in the positive direction of the tool axis, which means that a movement towards the work piece is in the positive direction. The positioning logic is changed accordingly.

SWITCHING OFF POSITIONING LOGIC

Sometimes the positioning logic is not required, e.g. when moving the tool to a tool change position. If the positioning logic is switched off, all axes move simultaneously.

The positioning logic can be switched off with the G28 function and the word I5=1. Refer to G28 for details.

PARALLEL AXES

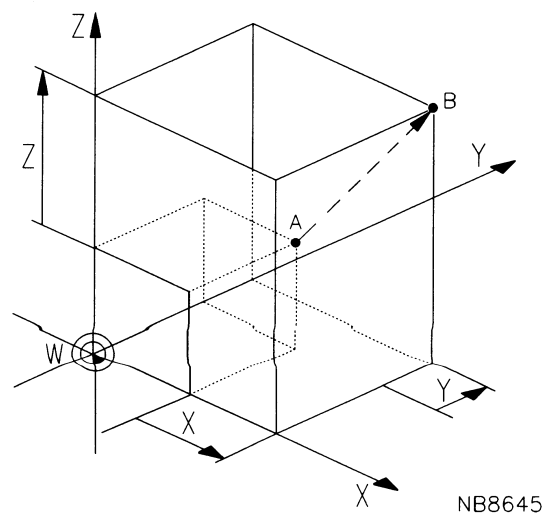
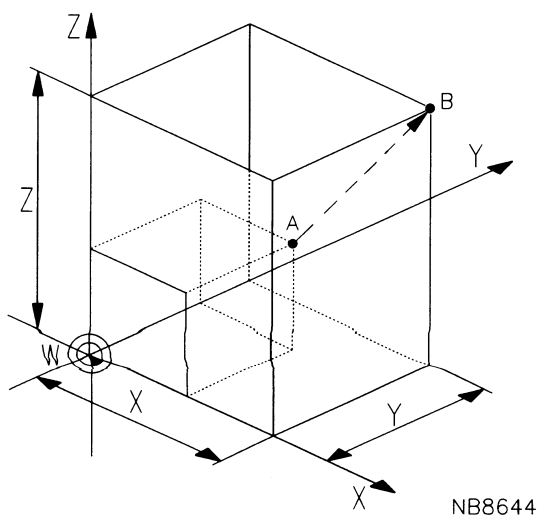
If available on the machine tool, the linear axes U, V and W, which are parallel to the main axes X, Y and Z, can be used instead of X, Y and Z. Only Cartesian coordinates can be used with the U, V and W axes.

ORIENTED SPINDLE STOP (D.. M19)

The D-word for the offset angle with oriented spindle stop must be programmed together with the function M19. Refer to the function M19 for details.

STARTING POSITION

At the beginning of a program, every active axis should be programmed in a program block for axis movements to ensure that each axis is in the starting position after a program start.

Example

The rapid movement from point A to point B can be programmed as:
N... G0 X25 Y15 Z30

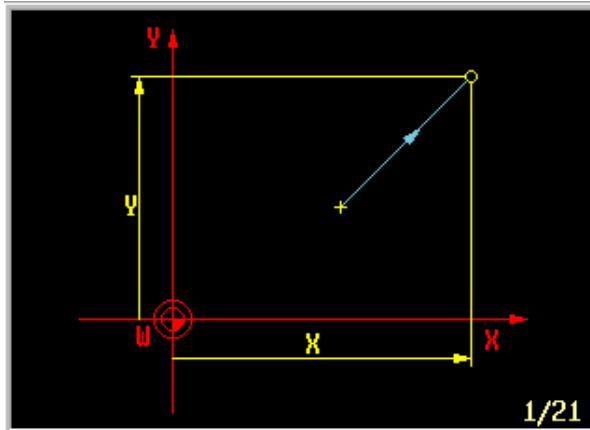
The actual movements are:

- a simultaneous movement in the main plane (X and Y);
- a movement in the tool axis (Z).

5.2 G1 Linear interpolation

Purpose

Commands all linear movements to be at specified feedrates.



Format

Main plane movements

N... G1 {X..}{Y..}{Z..}{F..}

3D-interpolation

N... G1 X.. Y.. Z.. {F..}

One rotary axis

N... G1 {A..}{B..}{C..}{A40=..}{B40=..}{C40=..}{F...}

Multi axes movements

N... G1 {X..}{Y..}{Z..}{A..}{B..}{C..}{A40=..}{B40=..}{C40=..}{F..}

Parameter

G Linear interpolation
 X Endpoint coordinate
 Y Endpoint coordinate
 Z Endpoint coordinate
 A Endpoint angle
 B Endpoint angle
 B1= Angle
 B2= Polar angle
 ?90= Endpoint abs. (X,Y,Z..)
 ?91= Endpoint incr. (X,Y,Z..)
 L1= Path length
 L2= Polar length
 P1= Point definition number
 P2= Point definition number

P3= Point definition number
 P4= Point definition number

Modal Words

A40=, B40=, C40=
 F, S, T, M, H, E..=
 F1=, T1=, T2=
 D (Angle oriented spindle stop)

Type of function

Modal

Associated functions

G0, G2/G3, G6
 F-functions
 Wordwise absolute/incremental programming (X90=.., X91=..) .

Notes and Usage**AXIS MOVEMENTS**

Axis movements are always interpolated, so that they occur simultaneously along all the programmed axes.

POLAR COORDINATES OR ONE COORDINATE AND ANGLE

Positions in the main plane can also be programmed with polar coordinates or one coordinate and angle.

DEFINED POINTS (G78)

A maximum of four previously defined points (P-words) can be stated in a G1 program block. The sequence in which the points are programmed also establishes the tool movement sequence e.g.:

G1 P5 P2 F100

Tool moves at 100 mm/min first to P5 and then to P2.

FEED RATE IN MAIN PLANE

The programmed feed rate is the feed on the straight line.

RADIUS COMPENSATION IN MAIN PLANE (G40 - G44)

Radius compensation on contours defined with linear and circular movements is available. Refer to the functions G40, G41/G42 and G43/G44 for additional information.

3D-INTERPOLATION

If the three axes X, Y and Z are programmed in one block, the CNC will control the axes movements so that a linear movement in space is made from the start point to the point defined by the three end point coordinates.

Positions in the main plane can be programmed normally.

For programming axes outside the main plane Cartesian coordinates must be used.

FEED RATE WITH 3D-INTERPOLATION

The programmed feed rate is the feed on the straight line.

3D TOOL CORRECTION

A 3D tool correction with normalized vectors is available. Refer to the G141 section for additional information.

PROGRAMMING ROTARY AXES

RADIUS OF ROTARY AXIS FOR FEED CALCULATIONS

For feed calculations the radius of each rotary axis involved can be programmed with A40= (for the A-axis), B40= (for the B-axis) and C40= (for the C-axis).

MACHINES WITH KINEMATIC MODEL

In machines with the kinematical model, the radius of the axis of rotation between the centre point of the rotary axis and of the workpiece can be calculated. This means that A40=, B40= and C40= no longer need to be programmed. This is enabled via G94 F5=1.

CANCELLATION OF THE RADIUS OF THE ROTARY AXIS

The programmed radius of the rotary axis is modal and, therefore, remains active until cancelled by:

- A40=0, B40=0 or C40=0.
- A different coordinate system being selected.
- M30, CLEAR CONTROL or CANCEL PROGRAM.

PROGRAMMED FEED RATE

When A40=, B40= or C40= is given a radius value, the surface feed is programmed in mm/min or inch/min.

NO RADIUS OF ROTARY AXIS PROGRAMMED

If no radii are programmed, the programmed feed rate is the feed of the path of the linear axes and is used by the CNC for calculating the feed for each rotary axis. This ensures that all axes cover the same part of their distances to go in the same time.

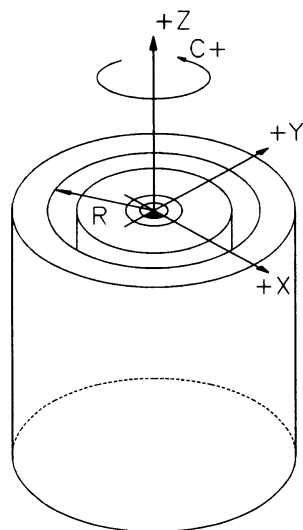
MAXIMUM FEED PER AXIS

If the maximum feed of an axis (MC-setting) is exceeded, the actual feed is reduced, so that the movement is performed with the maximum feedrate.

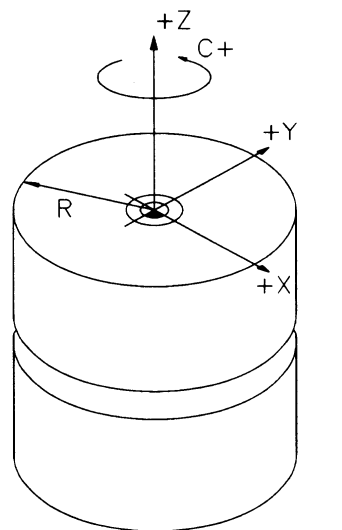
ONE ROTARY AXIS

Note: In the following example, the Z- and C-axis have been used, however, the same principles apply to the Y-/B-axis and X-/A-axis combinations.

With just the rotary axis moving, two cutting actions are possible.



1. Groove cut in the facing plane



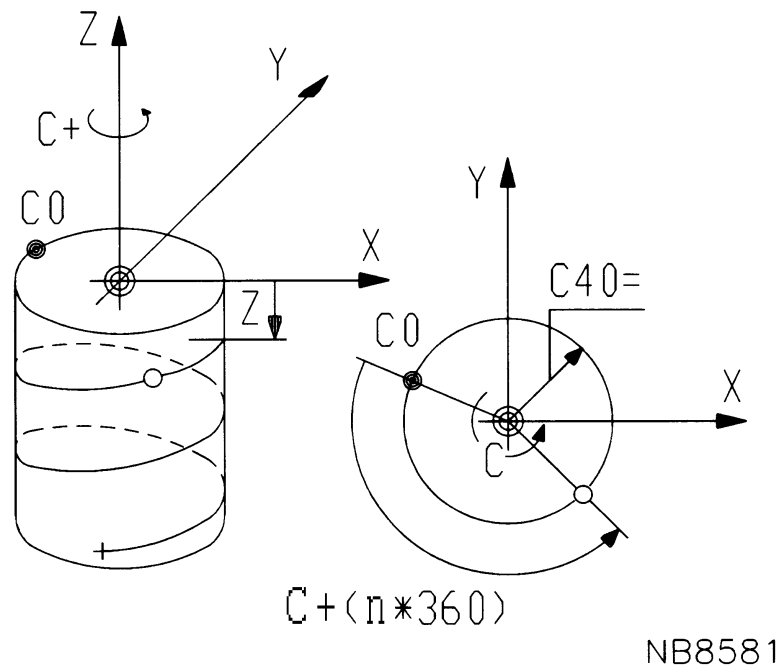
NB8569

2. Groove cut in the cylinder's surface

FEED RATE WITH ROTARY AXIS ONLY

If the radius of the rotary axis is programmed (A40=., B40=., C40=.), the feed rate is the surface feed in mm/min or inch/min.

If the radius of the rotary axis is not programmed or A40=0, B40=0 and C40=0 is programmed, the feed rate is the feed in degrees/min.

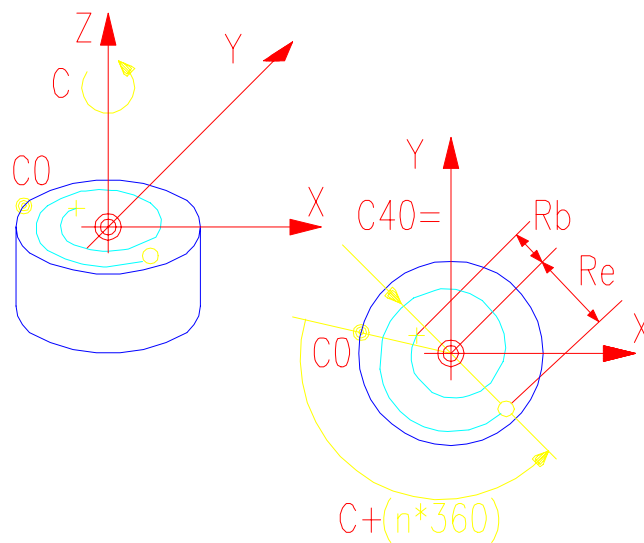
ONE ROTARY AXIS AND ONE LINEAR AXIS

Helix on the curved surface of a cylinder

CYLINDER INTERPOLATION

If contours or positions are to be programmed on the curved surface of a cylinder, the function G182 is available for an easy programming of these movements. Refer to the G180/G182 section for additional information.

SPIRAL IN THE FACING PLANE

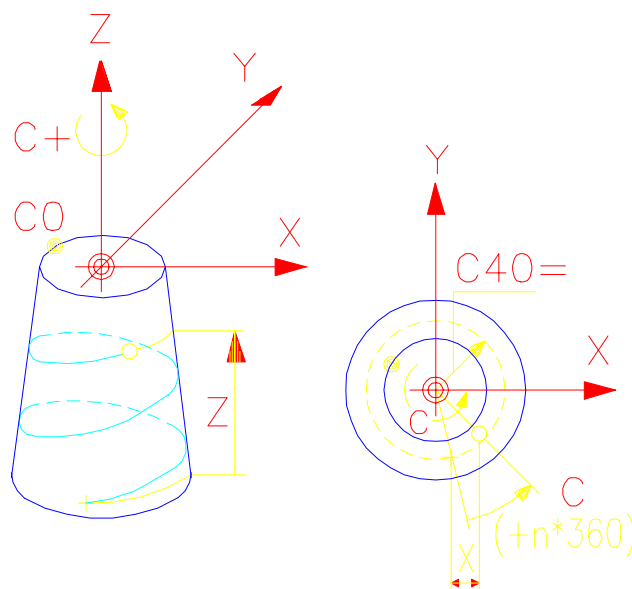


AVERAGE RADIUS OF THE SPIRAL

The radius value, which should be programmed for feed calculations, is the average radius of the path; this radius is used by the CNC to calculate the required feed rate to produce the spiral. The average radius (e.g. C40=) is calculated by using the formula:

$$C40 = (\text{radius at start point } (Rb) + \text{radius at end point } (Re)) / 2$$

ONE ROTARY AXIS AND TWO LINEAR AXES

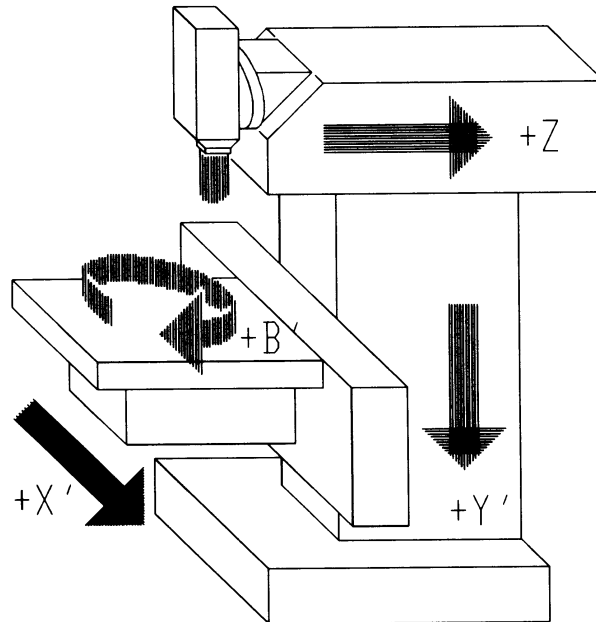


For feed calculations the average radius (C40=) has to be used and is calculated as:

$$C40 = (\text{radius at start point} + \text{radius at end point}) / 2$$

The movement on the cone is programmed with all three axes (X, Z, and C) in one block.

MULTI AXES PROGRAMMING



Machine tool with rotary table and tilting head

In a G1-block any combination of the three linear axes X, Y, or Z and the three rotary axes A, B or C (if available) is allowed.

Positions in the main plane can be programmed normally.

For programming axes outside the main plane Cartesian coordinates must be used.

GENERAL REMARKS WITH G1

PARALLEL AXES

If available on the machine tool, the linear axes U, V and W, which are parallel to the main axes X, Y and Z can be used instead of X, Y and Z. Only Cartesian coordinates can be used with the U, V and W axes.

CANCELLATION

The function G1 is cancelled by functions G1, G2/G3, G6 or at end of program (M30), CLEAR CONTROL or softkey CANCEL PROGRAM.

START NEXT MOVEMENT

In general feed movements are executed without a stop between the blocks. This results in rounded corners.

G28 and parameter I3= allows to program if the next movement starts after a full stop of the tool or without a stop between the movements.

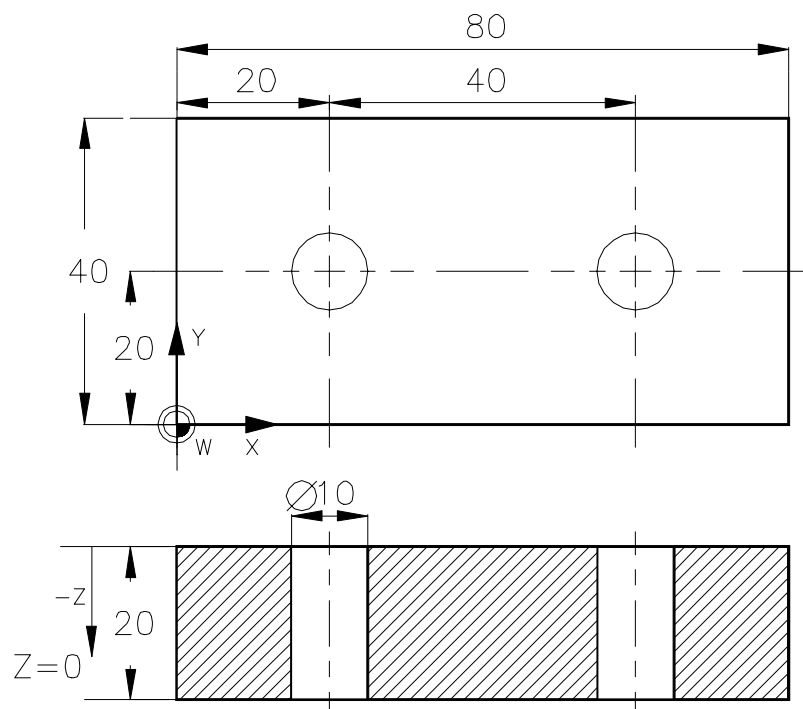
Refer to the function G28 for details.

ORIENTED SPINDLE STOP (D..M19)

The D-word for the offset angle with oriented spindle stop must be programmed together with the function M19. Refer to the function M19 for details.

Example

example 1. Drilling



```

N9001
N1 G17
N2 G54 T1 M6 S1000
N3 X20 Y20 Z1 F150 M3
N4 G1 Z-23.5
N5 G0 X60 Z1
N6 G1 Z-23.5
N7 G0 Z200 M30

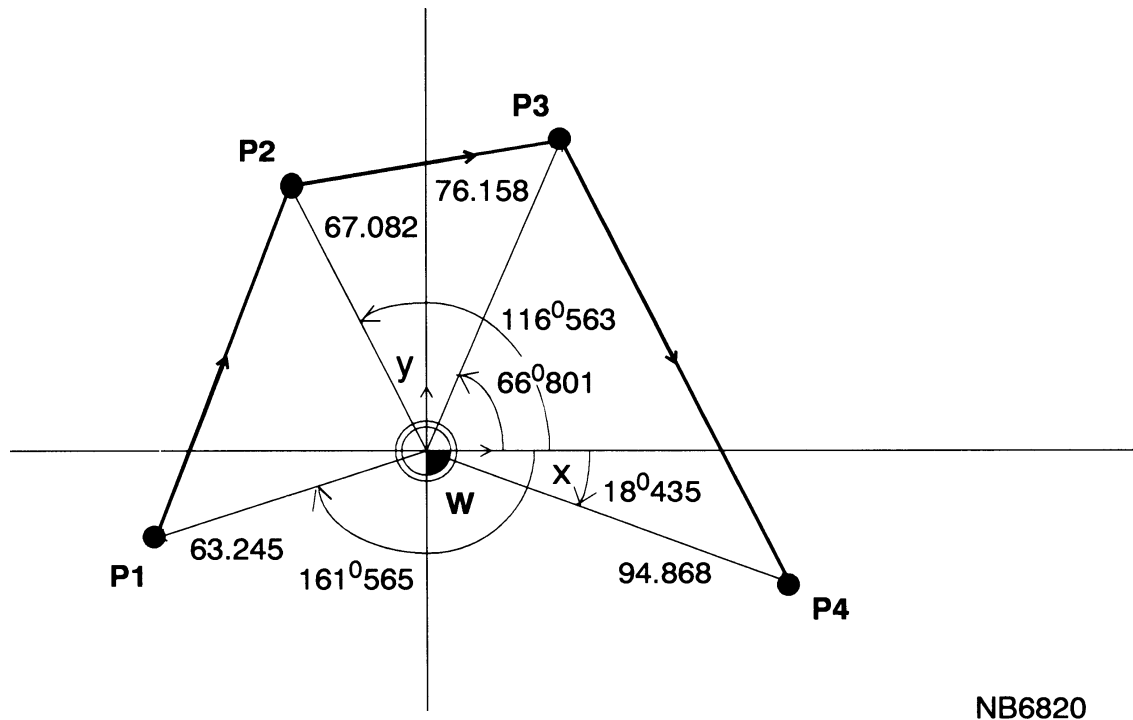
```

Explanation:

- N1: Activate XY-plane (G17).
- N2: Activate zero shift (G54). Load tool T1 and its offsets. Spindle rotation at 1000 rev/min. Drill diameter 10 mm.
- N3: Move tool rapidly (G0) to programmed position. Set feedrate to 150 mm/min. Make spindle rotate clockwise (M3).
- N4: Feed tool to programmed depth.

- N5: Retract tool to Z1 and then move the tool rapidly to X60. The CNC's positioning logic ensures that the tool does not collide with the workpiece, because the tool is first moved along the Z-axis before moving along the X-axis.
- N6: Feed tool to programmed depth.
- N7: Retract tool to Z200 and end of program.

EXAMPLE 2. Main plane movements



By using absolute polar coordinates the above movements can be programmed as follows:

```

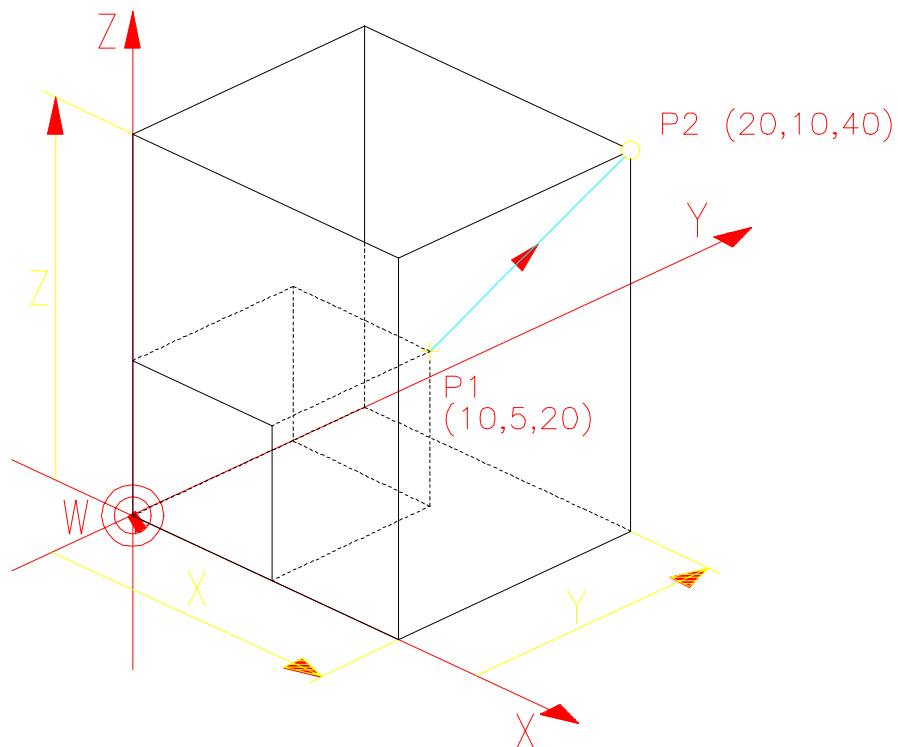
N10 G0 B2=-161.565 L2=63.245 F100 (P1)
N11 G1 B2=116.565 L2=67.082 (P2)
N12 B2=66.801 L2=76.158 (P3)
N13 B2=-18.435 L2=94.868 (P4)

```

Explanation:

- N10: Move at rapid traverse rate (G0) to point P1.
- N11-N13: Move tool at feedrate (100 mm/min) to the points P2, P3 and P4.

EXAMPLE 3. 3D-interpolation

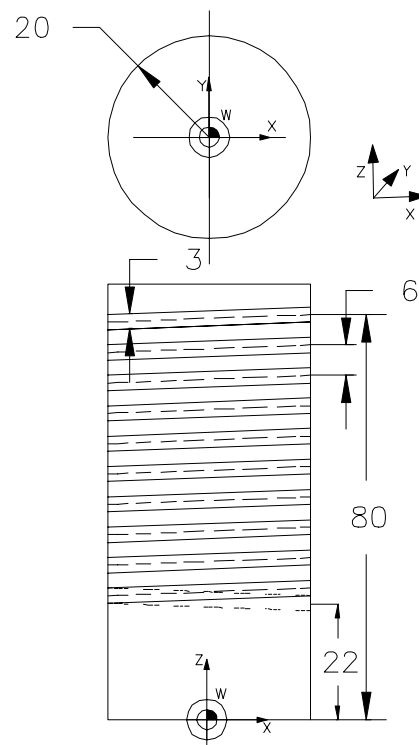


Tool moves from P1 (10,5,20) to P2 (30,10,40) at a feedrate of 100 mm/min

```
N14 G0 X10 Y5 Z20  
N15 G1 X20 Y10 Z40 F100
```

In block N15 the three axes move simultaneously and reach their end positions at the same time.

BEISPIEL 4. Spirale auf einer Zylinderfläche



Simultaneous movements of Z- and C-axes. The tool is in the Y-axis. The helix has 10 turns and a pitch of 6 mm.

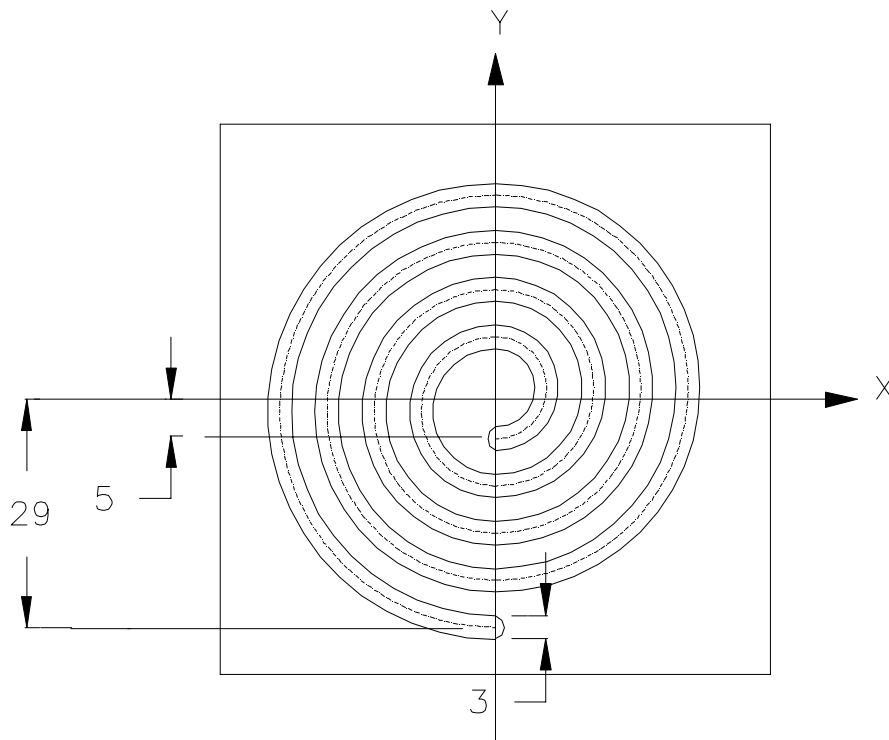
```

N10 G18
N11 T1 M6
N12 G0 X0 Y22 Z80 C0 S3000 M3
N13 G1 Y18 F75
N14 Z20 C3600 C40=20 F125
N15 G0 Y100
  
```

Explanation:

N10: Define the main plane
 N11: Load tool T1 and its offsets (mill diameter 3 mm). Tool is in Y-axis (G18).
 N12: Start spindle and move tool to start position.
 N13: Feed tool 4 mm to position Y18
 N14: Mill the helix. The rotary axis turns ten times (C3600)
 N15: Move tool away from the workpiece.

EXAMPLE 5. Spiral in the main plane (= facing plane) of a vertical milling machine.



The spiral has four turns and a pitch of 6 mm. It is produced with a rotary table (C-axis) rotating around the tool axis (Z-axis) and a simultaneous movement in the Y-axis. C40= 17

The program could look like:

```
N10 G17
N11 G54 T1 M6
N12 G0 X0 Y-5 Z2 C0 S3000 M3
N13 G1 Z-2 F75
N14 Y-29 C1440 C40=17 F200
N15 G0 Z100
```

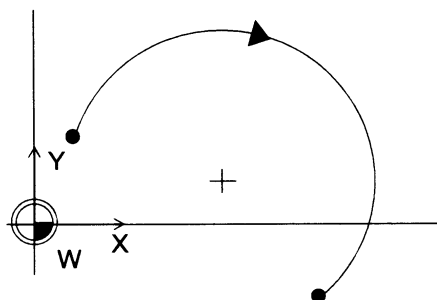
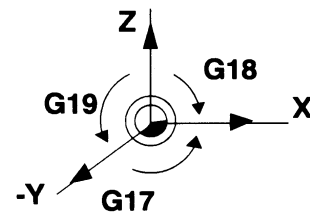
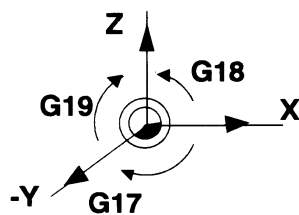
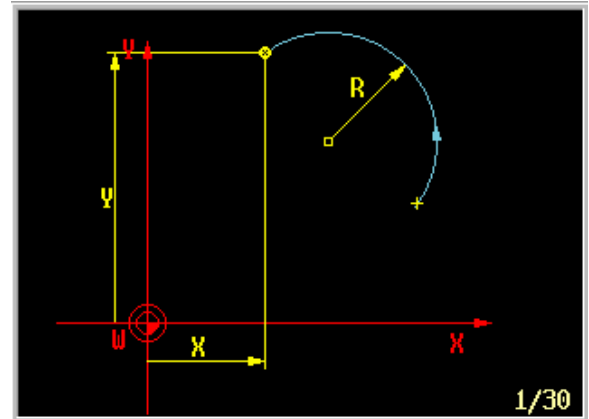
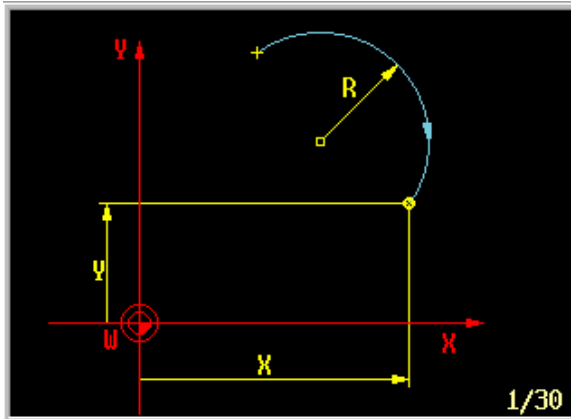
Explanation:

- N10: Define the main plane
- N11: Activate zero shift to the middle of the workpiece (G54). Load tool T1 and its offsets. The tool is in the Z-axis (G17). Mill diameter 3 mm.
- N12: Start the spindle. Move tool to start position.
- N13: Feed tool in Z-axis: 2 mm depth into the workpiece.
- N14: Mill the spiral (Rb=5, Re=29, therefore C40=17). Rotate the C-axis four times (C1440).
- N15: Move tool away from workpiece.

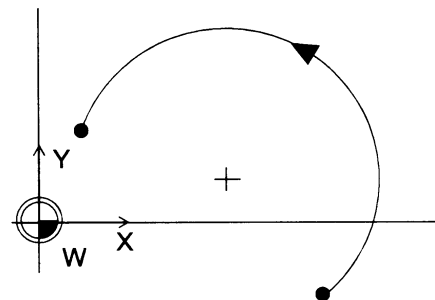
5.3 G2/G3 Circular interpolation (CW/CCW)

Purpose

To execute a Clockwise (G2) or Counter-Clockwise (G3) circular movement at a specified feedrate.



G2



G3

NB8710

The direction of circular movement is decided by looking in the negative direction of the tool axis towards the main plane.

Format

Full circle.

N... G2/G3

[Centre point coordinates]

Arc less than or equal to 180°:

N... G2/G3

[Linear axis's end point coordinates] R...

Arc less or greater than 180°:

N... G2/G3 [Centre point coordinates] [Linear axis's end point coordinates]
 N... G2/G3 [Centre point coordinates] B5=...

2.5D Interpolation

N... G2/G3 [Centre point coordinates] [Arc end point coordinates]
 [Linear or rotary axis's end point coordinate]

Helix

N... G2/G3 [Centre point coordinates] [Arc end point coordinates]
 [Linear or rotary axis's end point coordinates] [Pitch]
 N... G2/G3 [Centre point coordinates] [Pitch] B5=...

Parameter

G Circular clockwise
 X Endpoint coordinate
 Y Endpoint coordinate
 Z Endpoint coordinate
 A Endpoint angle
 B Endpoint angle
 I Center point in X / pitch in X
 J Center point in Y / pitch in Y
 K Center point in Z / pitch in Z
 R Circle radius
 B1= Angle
 B2= Polar angle
 B3= Polar angle for center
 B5= Angle of arc

?90= End-/centrep. abs. (X,Y,Z..I,J,K)
 ?91= End-/centrep. incr.(X,Y,Z..I,J,K)
 L1= Path length
 L2= Polar length
 L3= Polar length for center
 P1= Point definition number

End point coordinates

X, Y, Z Endpoint coordinate
 A, B, C Endpoint angle
 B1= Angle
 L1= Path length
 B2= Polar angle
 L2= Polar length
 P1= Point definition number

Center point coordinates

I Center point in X
 J Center point in Y
 K Center point Z / Pitch in Z
 B3= Polar angle for center
 L3= Polar length for center

Circle parameters

R Circle radius
 B5= Angle of arc
 D Angle oriented spindle stop

Modal Words

F, S, T, M, H, E..=
 F1=, T1=, T2=

Type of function

Modal

Associated functions

G0, G1, G6

F-functions

Wordwise absolute/incremental programming (X90=..., X91=..).

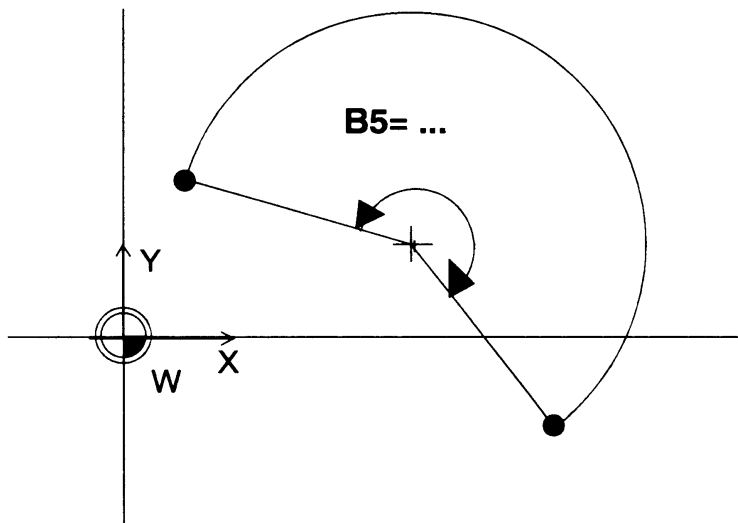
Notes and usage**CIRCLE IN THE MAIN PLANE****CIRCULAR ARC UP TO 180 DEGREES**

Using the end point's coordinates together with the arc radius or the coordinates of the arc's centre point programs an arc movement up to 180 degrees.

The arc radius is programmed with the R-word. This is a dimension word without a sign.

CIRCULAR ARC GREATER THAN 180 DEGREES

An arc movement greater than 180 degrees can only be programmed with the coordinates of the end point and of the arc's centre point.

ANGLE OF CIRCULAR ARC (B5=)**NB8714**

An arc of any angle between 0° and 360° can also be programmed with the centre point coordinates and the angle of the arc. The angle is programmed with the word B5= in decimal degrees and without sign.

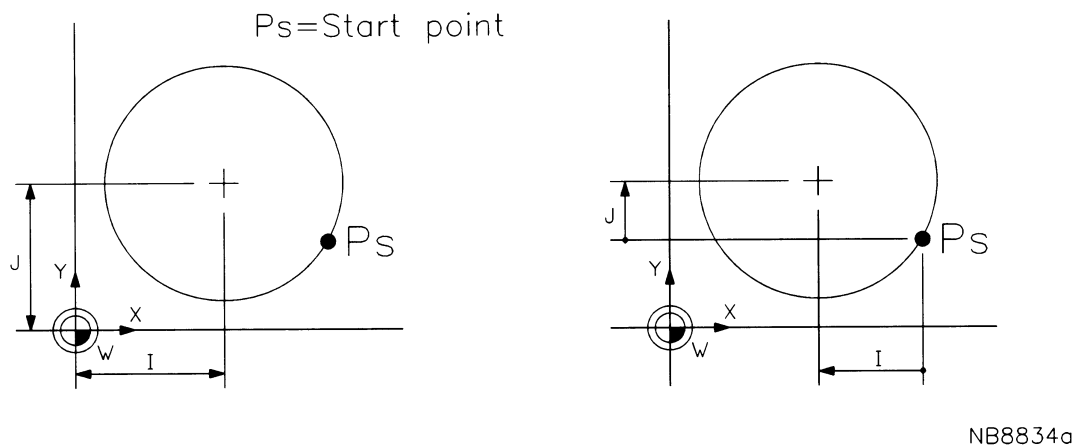
PROGRAMMING A COMPLETE CIRCLE

After making a complete circle the tool is back at its start point. This movement is programmed with:

- the direction of movement on the circle,
- the coordinates of the centre point.

Note: If the end point coordinates are also programmed in the block, no circular movement will be executed.

CENTRE POINT COORDINATES



Absolute (G90) coordinates

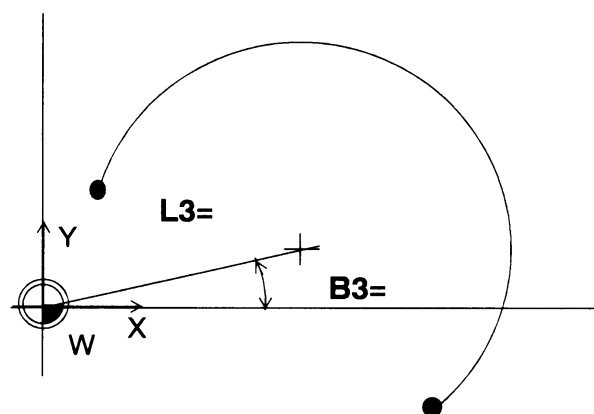
Incremental (G91) coordinates

Absolute centre point coordinates are related to the program zero point W.
Incremental centre point coordinates are measured from start point to centre point.

CENTRE POINT COORDINATES IN DIFFERENT PLANES

N... G2/G3	I	(X-axis)	J	(Y-axis): XY-PLANE (G17)
N... G2/G3	I	(X-axis)	K	(Z-axis): XZ-PLANE (G18)
N... G2/G3	J	(Y-axis)	K	(Z-axis): YZ-PLANE (G19)

POLAR CENTRE POINT COORDINATES

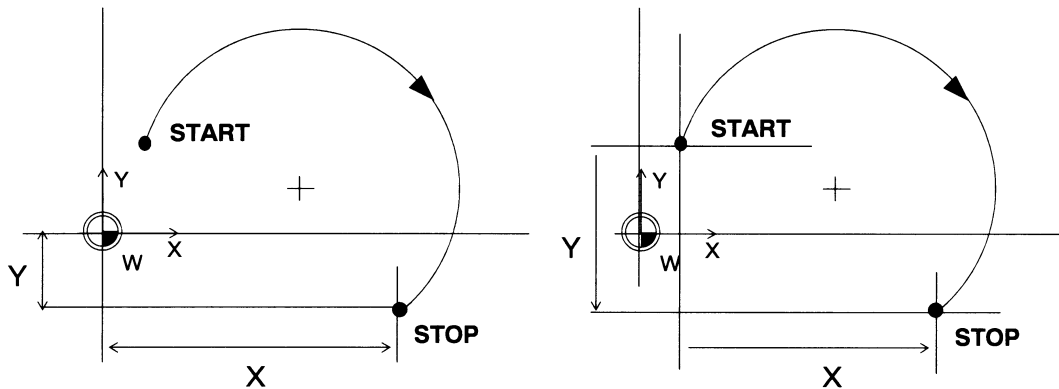


NB8711

L3=... B3=

The polar centre point coordinates are used in the plane defined by G17, G18 or G19.

CARTESIAN END POINT COORDINATES



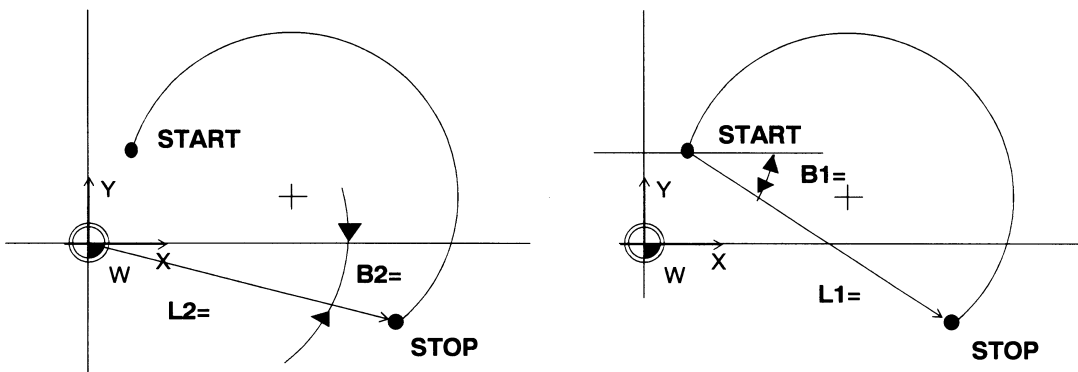
NB8712

Absolute (G90) coordinates

Incremental (G91) coordinates

(G17) X... Y... (G18) X... Z... (G19) Y... Z...

POLAR END POINT COORDINATES AND ONE COORDINATE AND ANGLE



NB8713

B2=... L2=...

B1=... L1=...

These end point coordinates are used in the plane defined by G17, G18 or G19.

DEFINED POINT (G78)

A previously defined point (P-word) can be used to program the end point of a circular movement.

FEED RATE IN MAIN PLANE

The programmed feed rate is the feed on the circle.

RADIUS COMPENSATION IN MAIN PLANE (G40 - G44)

Radius compensation on contours defined with linear and circular movements in the main plane is available. Refer to the functions G40, G41/G42 and G43/G44 for additional information.

A correction of the feed rate with circular movements and radius compensation depending on the shape of the contour and the radius of the mill is available. Refer to CONSTANT CUTTING FEED with the function G41/G42 for additional information.

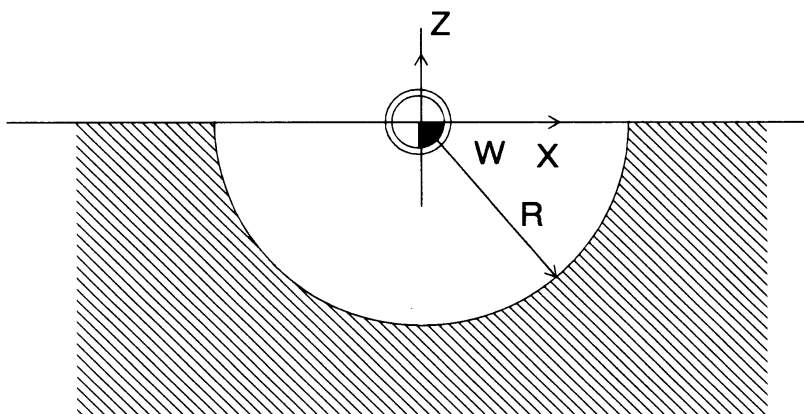
AXIS ROTATION (G92/G93 B4=)

A circular movement in a rotated main plane can be used.

A CIRCULAR MOVEMENT NOT IN THE MAIN PLANE

It is possible for a special tool such as a ball cutter, to be controlled so that it cuts in a direction being not parallel to the main plane. In these circumstances only Cartesian absolute or incremental dimensions can be used to program the end point and centre point coordinates.

Radius compensation is not available.

**NB8574**

Circular movement in XZ-plane; tool in Z-axis

CIRCULAR ARC UP TO 180°

An arc movement up to 180 degrees is programmed by using the end point's coordinates together with the arc radius (R-word) or the Cartesian coordinates of the arc's centre point.

CIRCULAR ARC GREATER THAN 180°

An arc movement greater than 180 degrees can only be programmed with the Cartesian coordinates of the end point and the arc's centre point.

ADDRESSES FOR END POINT AND CENTRE POINT

The addresses given in the table under the heading CENTRE POINT COORDINATES IN DIFFERENT PLANES also apply to arc movements, which are not in the current main plane. The addresses of the centre point define the plane in which the circle is to be milled.

AXIS ROTATION (G92/G93 B4=)

A circular arc is not in the plane defined by the active G-function for plane selection (G17, G18 or G19) is not allowed in a plane of which one axis is rotated. E.g. if G17 is active and the XY-plane rotated 30°, the X-axis is rotated. So a circular movement in the XZ-plane is not allowed. An error message is displayed to this effect.

LINEAR AXES U, V AND W

If a machine tool is equipped with linear axes parallel to the main axes, circular movements can be used with these axes.

Polar coordinates or radius compensation cannot be used.

PROGRAMMING A CIRCULAR ARC

For an arc movement up to 180 degrees, either the radius (R-word) or the Cartesian centre point coordinates (absolute or incremental) can be programmed.

For an arc movement greater than 180 degrees only Cartesian coordinates can be used for programming the centre point.

PROGRAMMING END POINT AND CENTRE POINT

Both coordinates of the endpoint must be programmed because they determine the plane in which the circular movement occurs.

Centre point coordinates are defined by: I for the U-axis; J for the V-axis; K for the W-axis. The tables below indicate which addresses are used with different planes.

Main axis X and a linear axis

	XV-plane	XW-plane
End point	X and V	X and W
Centre point	I and J	I and K

Main axis Y and a linear axis

	YU-plane	YW-plane
End point	Y and U	Y and W
Centre point	J and I	J and K

Main axis Z and a linear axis

	ZU-plane	ZV-plane
End point	Z and X	Z and Y
Centre point	K and I	K and J

Combination of linear axes

	UV-plane	UW-plane	VW-plane
End point	U and V	U and W	V and W
Centre point	I and J	I and K	J and K

A CIRCULAR MOVEMENT WITH A SIMULTANEOUS MOVEMENT IN A THIRD AXIS

The CNC control can use a special interpolation procedure (2.5D), to coordinate a circular movement in a main plane and a third axis' movement, so that a tool travels the correct paths from the start point to the end point.

CIRCLE IN THE MAIN PLANE

The normal programming methods for a circle in the main plane defined by G17, G18 or G19 are used.

Radius compensation with the circular movement can be used. The tool must tangentially enter and leave the workpiece.

If the third axis is the tool axis, using one of the addresses given in the table below programs it.

	G17	G18	G19
Plane	XY-plane	XZ-plane	YZ-plane
Tool axis	Z	Y	X

CIRCLE NOT IN THE MAIN PLANE

When the circular movement is not in the main plane, the rules given in A CIRCULAR MOVEMENT NOT IN THE MAIN PLANE must be used for programming the movement.

The Cartesian coordinates of the centre point define the plane. An arc radius (R-word) cannot be used.

The table below lists the addresses, which are used for different planes.

	XY-plane	XZ-plane	YZ-plane
End point	X and Y	X and Z	Y and Z
Centre point	I and J	I and K	J and K
Tool axis	Z	Y	X

THIRD AXIS IS A ROTARY AXIS

If the circular movement is executed in the main plane or in another plane as well, the third axis is not restricted to the tool axis, but a rotary axis programmed with the address A or B can also be used. In this case a simultaneous movement of the linear axes performing the circular movement in the defined plane, and the rotary axis occur.

HELIX INTERPOLATION

Programming the following can mill a helix on any cylinder surface:

- circular movement in the main plane as described
- the pitch of the helix
- (if necessary) the end point of the linear movement.

	G17	G18	G19
Tool axis	Z	Y	X
Centre point	I and J	I and K	J and K
	or	or	or
	B3= and L3=	B3= and L3=	B3= and L3=
Angle of arc	B5=	B5=	B5=
Pitch of helix	K	J	I

PROGRAMMING THE ARC ANGLE

The value of 'B5=' can be from 0 to 999999 degrees, which is approximately 900 revolutions.

PROGRAMMING THE TOOL AXIS

A helix movement can also be programmed by using the addresses given in the table below.

	G17	G18	G19
Tool axis	Z	Y	X
Circle end point	X and Y	X and Z	Y and Z
Centre point	I and J	I and K	J and K
Pitch of helix	K	J	I

When these alternative addresses are used, the movements have to be programmed so that the circular movement and the tool axis movement reach their end positions at the same time.

THIRD AXIS IS A ROTARY AXIS

The third axis is not restricted to the tool axis, but a rotary axis programmed with the address A or B can also be used. In this case a simultaneous movement of the linear axes performing the circular movement, and the rotary axis occur.

RADIUS COMPENSATION WITH HELIX INTERPOLATION

Radius compensation with the circular movement can be used during helix interpolation. The tool must tangentially enter and leave the workpiece.

GENERAL REMARKS WITH G2/G3

CANCELLATION

The function G2 or G3 is cancelled by any other function of group A, or at end of program (M30), or by CLEAR CONTROL.

CHECKING OF CENTRE POINT COORDINATES

When centre point coordinates are used, the radius of the circular movement at the start is compared with the radius at the end. If the difference between the two values is greater than a Machine Constant setting, the CNC generates an error message and stops program execution.

START NEXT MOVEMENT (G28 I3=..)

In general feed movements are executed without a stop between the blocks. This results in rounded corners.

G28 and parameter I3= allows to program if the next movement starts after a full stop of the tool or without a stop between the movements.

Refer to the function G28 for details.

FEED LIMITATION (G28 I6=..)

With feed limitation the programmed feed rate is reduced to keep the axes following error to an acceptable minimum and therefore to improve the machining accuracy. Refer to the function G28 I6= for details.

ORIENTED SPINDLE STOP (D.. M19)

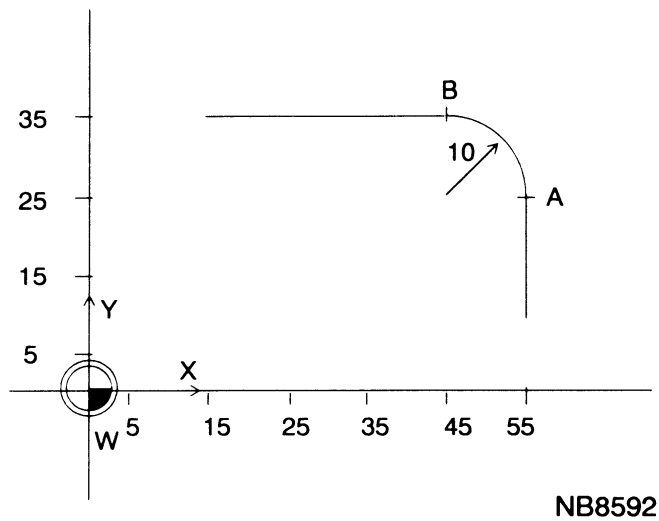
The D-word for the offset angle with oriented spindle stop must be programmed together with the function M19. Refer to the function M19 for details.

Note: If a circle is programmed and one of its endpoints is within some microns from the startpoint, then at high feed values the cnc will not perform a circle but a linear movement directly to the endposition.

Example: if difference between start and end position is 5 micron, then if the feed exceeds 1 m/ min then a linear movement will be executed although G2/G3 has been programmed.

Examples

EXAMPLE 1. Programming an arc radius

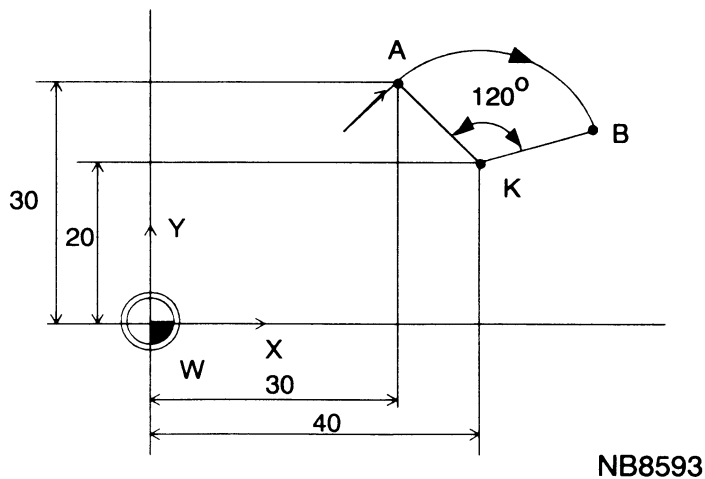


N10 G1 X55 Y25 F100 (A)
 N20 G3 X45 Y35 R10 (B)

Explanation:

N10: Move tool at set feedrate to the starting point A of the arc.
 N20: Move tool in a counter-clockwise direction (G3) to end point B.

EXAMPLE 2. Programming an arc angle.

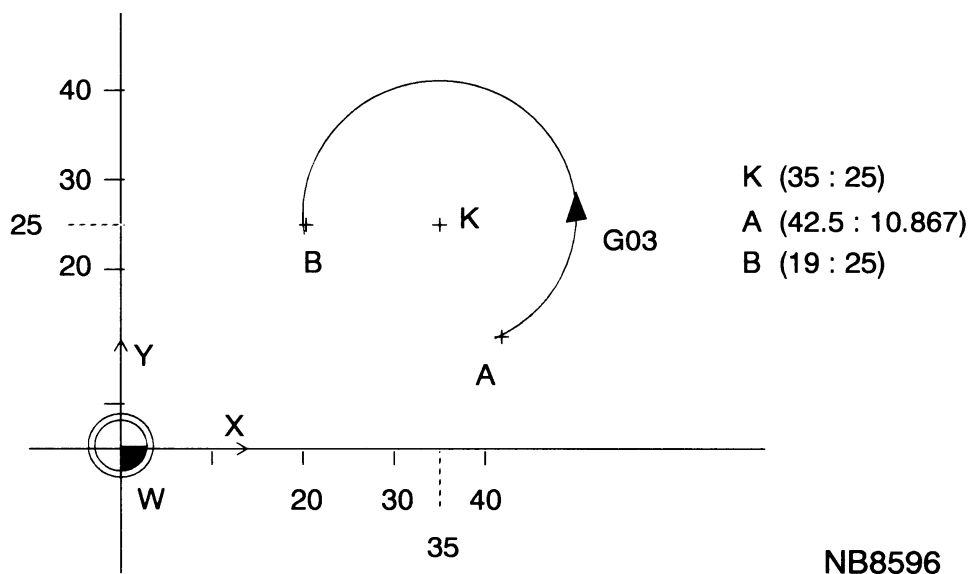


N10 G1 X30 Y30 F500 (A)
 N11 G2 I40 J20 B5=120 (B)

Explanation:

N10: Move tool at set feedrate to the starting point A.
 N11: Move tool in a clockwise direction (G2) to end point: the angle of the arc is stated by B5=.

EXAMPLE 3. Programming an arc > 180 degrees (Cartesian)



Absolute coordinates

```
N10 G1 X42.5 Y10.867 F200 (A)
N11 G3 X19 Y25 I35 J25 (B)
```

Explanation:

N10: Move tool at given feedrate to the starting point A of the arc.

N11: Move tool in a counter-clockwise (G3) direction to end point B. Centre point coordinates are stated by I and J.

Both the coordinates X and Y as well as I and J are absolute values with regard to the program zero point W.

Incremental coordinates

```
N10 G1 X42.5 Y10.867 F200 (A)
N11 G91
N12 G3 X-23.5 Y14.133 I-7.5 J14.133 (B)
```

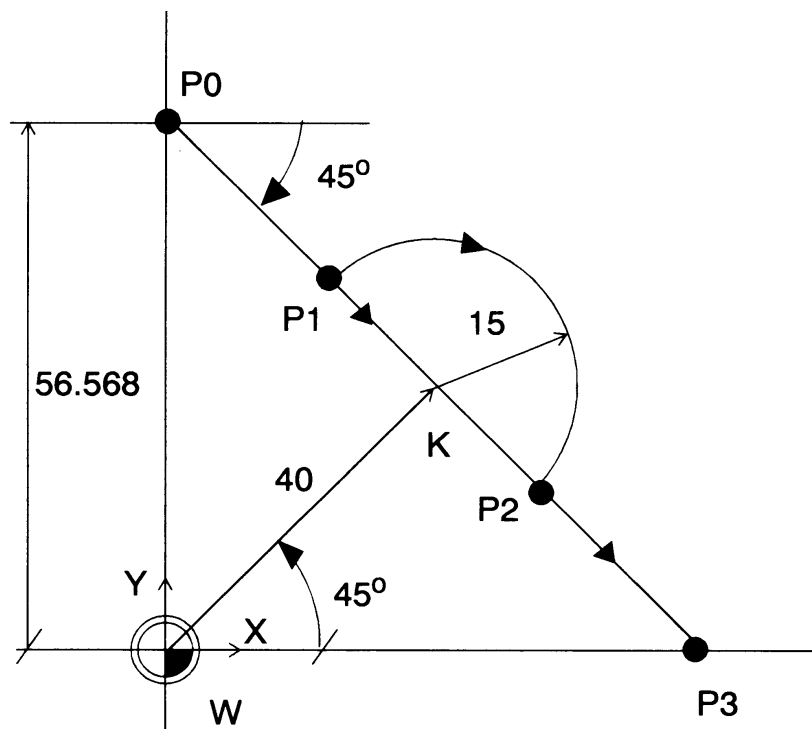
Explanation:

N10: Move tool at given feedrate to the starting point A of the arc.

N11: Activate incremental coordinate mode (G91).

N12: Move tool in a counter-clockwise (G3) direction to end point B. Coordinates X and Y are increments from point A to B. The coordinates I and J are incremental values from A to the centre.

EXAMPLE 4. Programming an arc with polar coordinates



NB8576

```

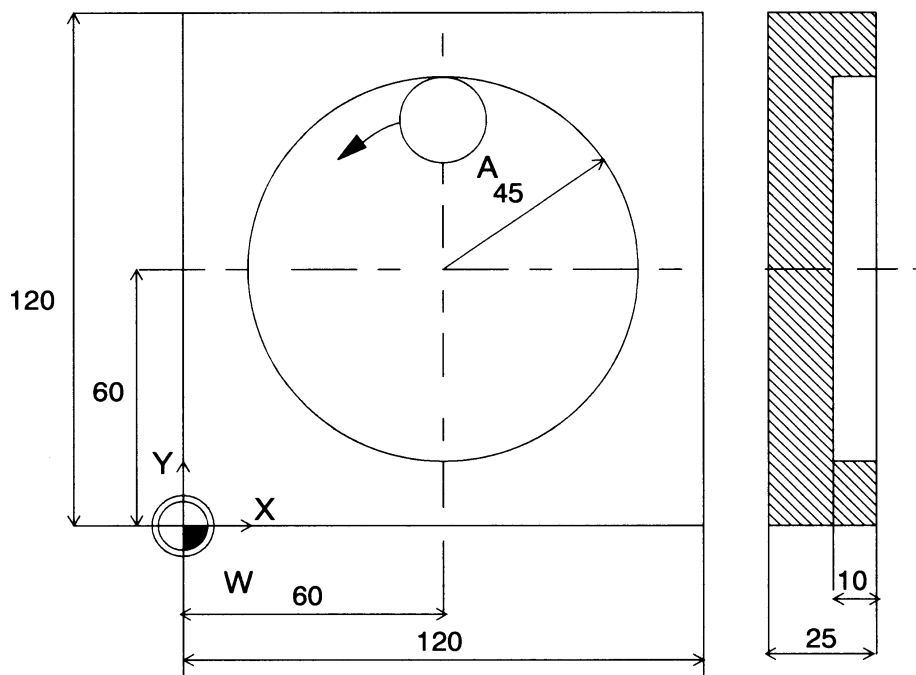
:
N10 G0 X0 Y56.568          (P0)
N11 G1 B1=-45 L1=25 F200   (P1)
N12 G2 B1=-45 L1=30 B3=45 L3=40 (P2)
N13 G1 B1=-45 L1=25        (P3)
:

```

Explanation:

- N10: Move tool rapidly to point P0.
- N11: Move tool at programmed feedrate from P0 and P1. Incremental polar coordinates are used.
- N12: Move tool in a clockwise (G2) direction from P1 to P2. For P2, incremental coordinates are used. The centre point is programmed with absolute polar coordinates (B3=, L3=).
- N13: Move tool at set feedrate from P2 to P3.

EXAMPLE 5. Programming a complete circular movement.



NB8575

```

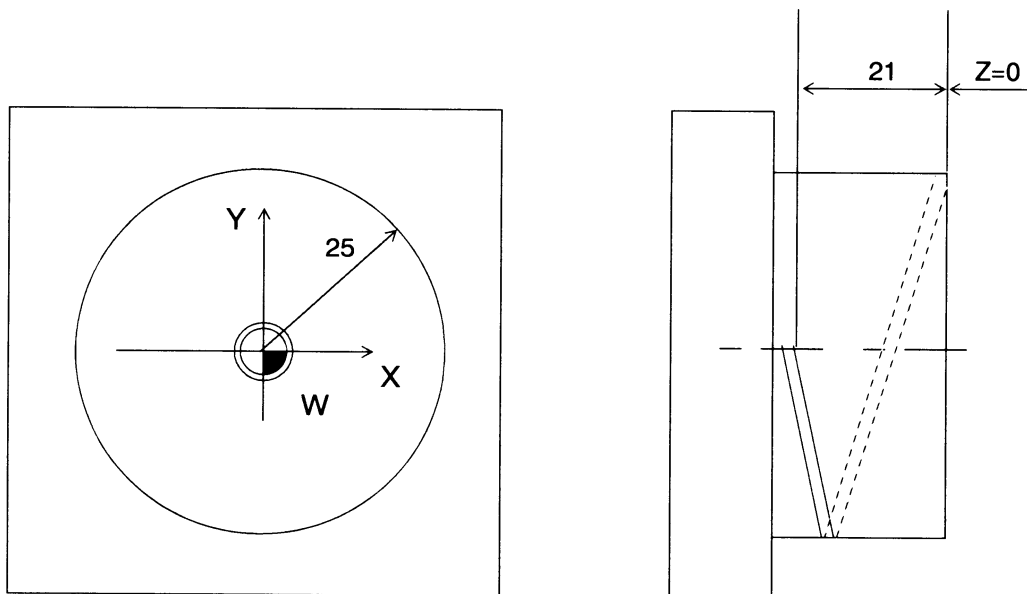
:
N9 G17 T1 M6
N10 G0 X60 Y60 Z10 S315 M3
N11 G1 Z-10 F36
N12 G43 Y105 F65
N13 G41
N14 G3 I60 J60
N15 G40
N16 G1 Y90
N17 Z100 M30
:

```

Explanation:

- N9: Activate XY-plane. Load tool T1 and its offset. (Mill diameter 16 mm)
- N10: Start the spindle and move tool rapidly to the centre of the pocket (Z10)(where the tool is to enter the hole). The feedrate is 36 mm/min
- N11: Set the feedrate to 65 mm/min and feed tool depth.
- N12: Move tool with feed to the wall (G43). To point A
- N13: Set radius compensation for a tool moving on the left handside (G41).
- N14: Mill the complete circle in a counter clockwise direction (G3).
- N15: Cancel radius compensation (G40).
- N16: Move tool away from milled surface of workpiece.
- N17: Retract tool out of workpiece.

EXAMPLE 6. Programming a circular movement together with a simultaneous linear axis movement (2.5D Interpolation).



NB8577

```

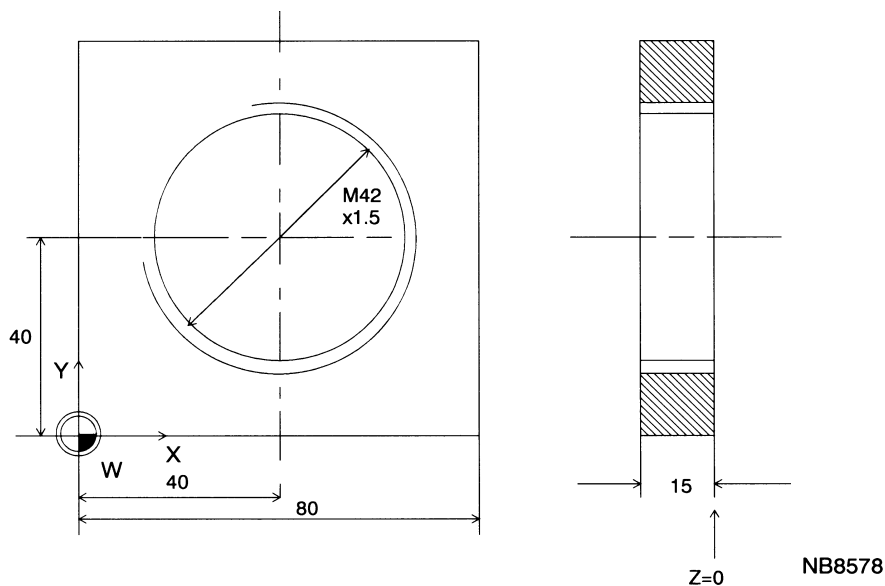
:
N10 G17 T1 M6
N11 G0 X0 Y35 Z0 S1000 M3
N12 G43
N13 G1 Y25 F80
N14 G41 F120
N15 G2 X-25 Y0 Z-21 I0 J0
N16 G40
N17 G1 X-35
:

```

Explanation:

- N10: Activate XY-plane. Select tool 1 (diameter 3 mm) and its offsets.
- N11: Start the spindle and move the tool to the workpiece at 3000 rev/min
- N12: Activate radius compensation to the endpoint
- N13: Move tool to the workpiece contour. Set linear feedrate to 80 mm/min.
- N14: Set radius compensation LEFT.
- N15: Perform the simultaneous movement of the circle and third axis.
- N16: Cancel radius compensation.
- N17: Move tool away from workpiece.

EXAMPLE 7. Programming a helix.



```

N10 G17
N11 T1 M6
N12 G0 X40 Y40 Z1.5 S400 M3
N13 G1
N14 G43 Y61 F120
N15 G42
N16 G2 I40 J40 K1.5 B5=4320
N17 G40
N18 G1 Y40
N19 G0 Z100

```

Explanation:

N10: Define the main plane
 N11: Load tool 1 and its offsets. Spindle rotation 400 rev/min (Thread mill cutter)
 N12: Start the spindle and move tool to starting position.
 N13: Set linear feed movement
 N14: Move tool with feed to (G43) the part.
 N15: Set radius compensation RIGHT (G42).
 N16: Mill the helix. Programmed arc:
 - circle centre (I and J)
 - angle of the arc (B5=) 12 turns of 360 degrees
 - pitch of thread (K).
 N17: Cancel radius compensation (G40).
 N18: Move tool away from the wall.
 N19: Retract the tool.

By using an alternative set of addresses, block 16 could be re-written as:

```
N16 G2 X40 Y61 Z-16.5 I40 J40 K1.5
```

Programmed arc:

- circle end point (X and Y)
 - depth (Z)
 - circle centre (I and J)
 - pitch of thread (K).

5.4 G4 Dwell time

Purpose

To insert a dwell period in the execution of a program.

Format

N... G4 X...

Minimum dwell period: 0,1 Seconds

Maximum dwell period: 900 Seconds (15 Minutes).

Parameter

```
G    Dwell time
X    Dwell time in sec. (0.1-983)
```

Modal Parameter

S, T, M, H, E..=

T1=, T2=

Type of function

Non-modal

Notes and usage

MODAL PARAMETER

It is advised to program this function in a separate block without modal parameters.

If modal parameters are used in a dwell block, they are executed before the dwell.

Those M-functions, which are executed after the commands in a block, are also executed after the dwell.

EXECUTION

After the dwell the machining is continued as normal.

Example

N50 G4 X2.5

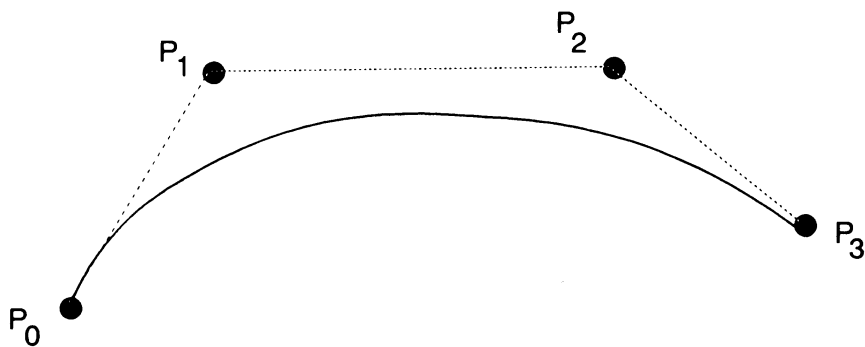
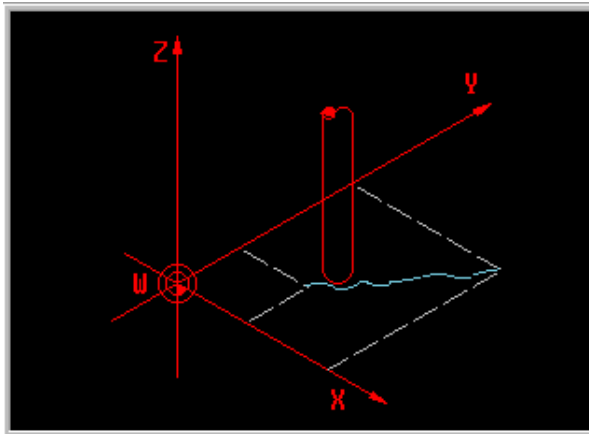
The above block causes a dwell of 2.5 Seconds between two operations

5.5 G6 Spline-interpolation

Purpose

The spline interpolation of the CNC enables the partprogrammer to input a series of points and have the control fit a smoothly faired curve through them.

By using this function, machine dynamic response is improved and leads to smoother tool movements and improved machining accuracy.

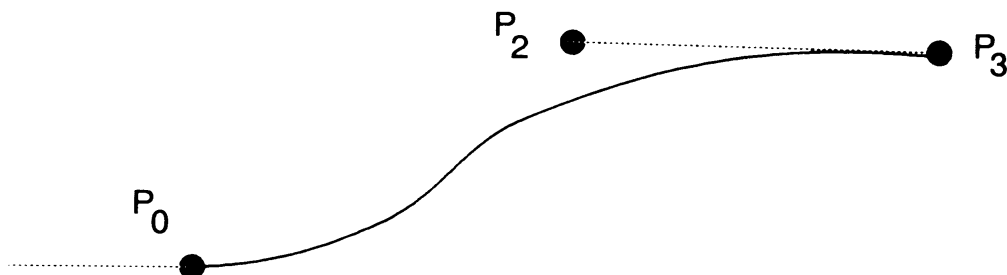


NB9771

Formats with Bezier splines

Spline with three vertex points

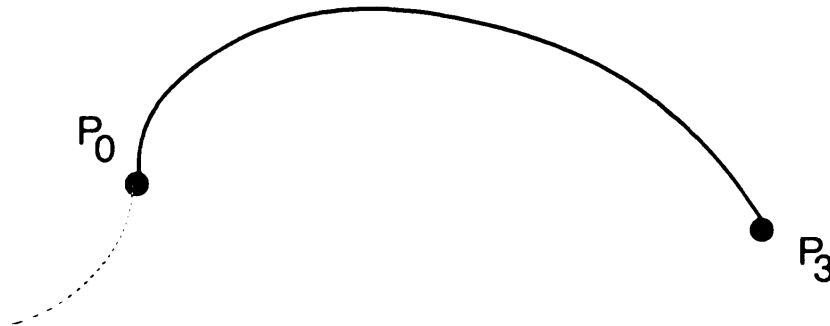
G6 X61=.. Y61=.. Z61=.. X62=.. Y62=.. Z62=.. X.. Y.. Z..



NB9772

Spline with two vertex points and constant tangent with previous spline

G6 X62=.. Y62=.. Z62=.. X.. Y.. Z..



NB9769

Spline with constant curvature with previous spline

G6 X.. Y.. Z..

Formats with cubic splines

Spline with all coefficients defined

G6 X51=.. Y51=.. Z51=.. X52=.. Y52=.. Z52=.. X53=.. Y53=.. Z53=..

Spline with constant tangent with previous spline

G6 X52=.. Y52=.. Z52=.. X53=.. Y53=.. Z53=..

Spline with constant curvature with previous spline

G6 X53=.. Y53=.. Z53=..

Parameter

G Spline interpolation
 X Endpoint (X-axis)
 Y Endpoint (Y-axis)
 Z Endpoint (Z-axis)
 X51= First spline coefficient
 Y51= First spline coefficient
 Z51= First spline coefficient
 X52= Second spline coefficient
 Y52= Second spline coefficient
 Z52= Second spline coefficient
 X53= Third spline coefficient
 Y53= Third spline coefficient
 Z53= Third spline coefficient
 X61= First support point (X-axis)

Y61= First support point (Y-axis)
 Z61= First support point (Z-axis)
 X62= Second support point (X-axis)
 Y62= Second support point (Y-axis)
 Z62= Second support point (Z-axis)

Bezier Splines

X, Y, Z	Endpoint (Z-axis)
X61=, Y61=, Z61=	First support point (Y-axis)
X62=, Y62=, Z62=	Second support point (Y-axis)

Cubic Splines

X51=, Y51=, Z51=	First spline coefficient
X52=, Y52=, Z52=	Second spline coefficient
X53=, Y53=, Z53=	Third spline coefficient

Modal functions

F, M

Type of function

Modal

Associated functions

G0, G1, G2/G3

F-functions

Notes and usage with Bezier splines**DEFINITION BEZIER SPLINES**

A Bezier spline is a spline defined by four points, the so-called vertex points.

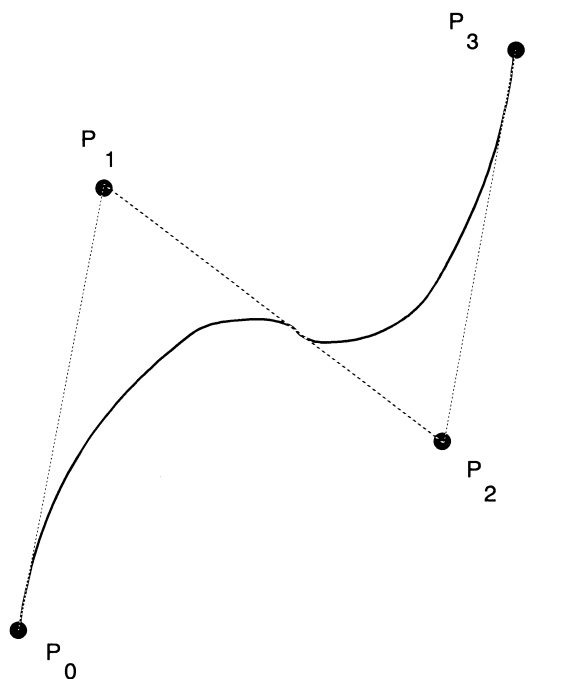
The first point of the spline is the end point of the previous movement. The curve passes through this point.

The first and second vertex points control the shape of the spline. The curve does not pass through these points.

The last point is the end point of the spline and this is, like the first point, a point through which the curve passes.

COORDINATES OF THE VERTEX POINTS

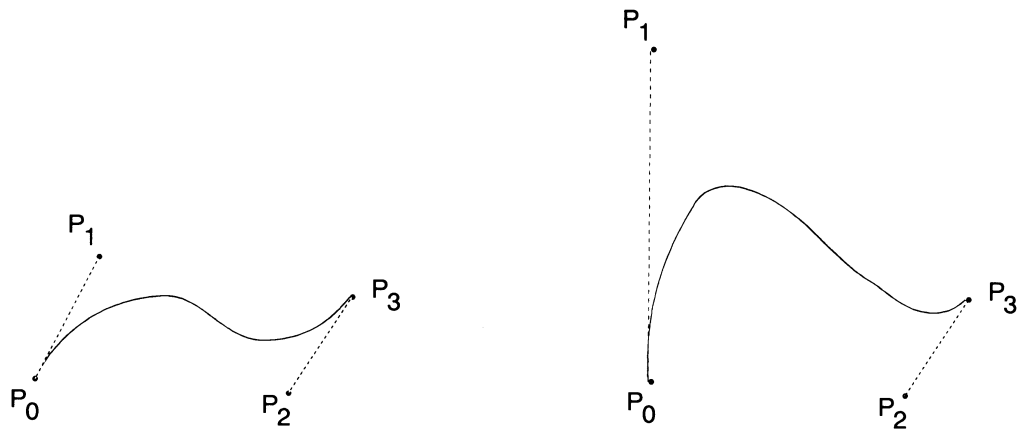
Only absolute Cartesian coordinates of the main axes X, Y and Z can be used for programming the vertex points. The coordinates are related to the program zero point. The coordinates of all three axes must be entered, even if not changed.

TANGENT AT START OF SPLINE

NB9764

From the coordinates of the first vertex point ($X_{61}=$, $Y_{61}=$, $Z_{61}=$) the tangent at the start of the spline is calculated as the line through the end point of the previous spline and the first vertex point.

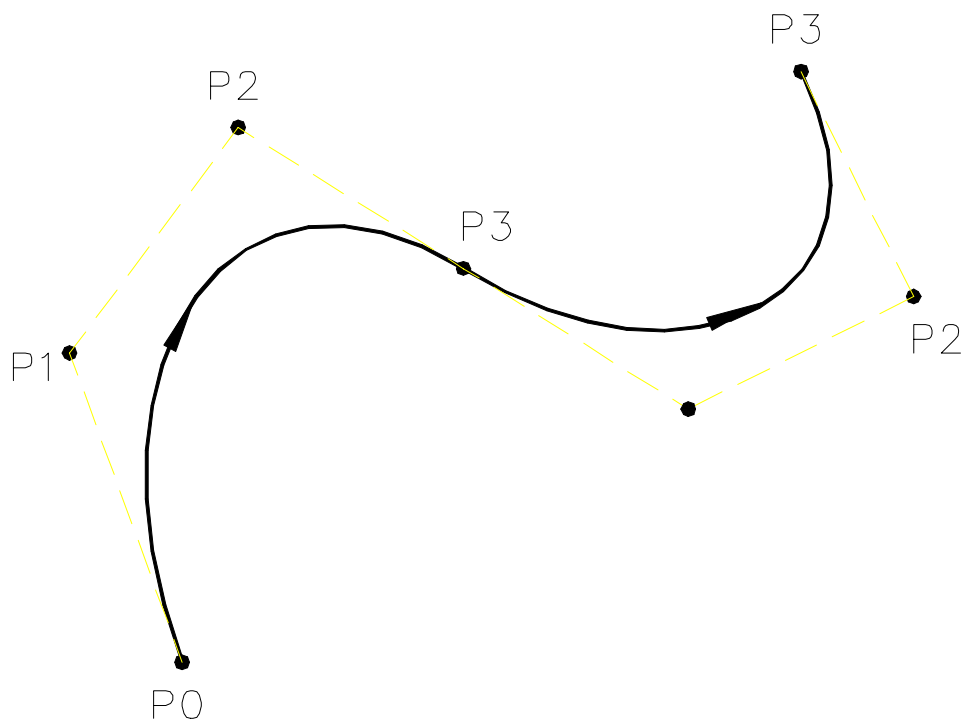
CHANGING THE LOCATION OF THE FIRST VERTEX POINT



NB9765

In the illustration three points of the spline are fixed and just the location of the first vertex point is changed. Notice that the shape of the curve changes.

TANGENTIAL CONTINUITY AT START OF SPLINE

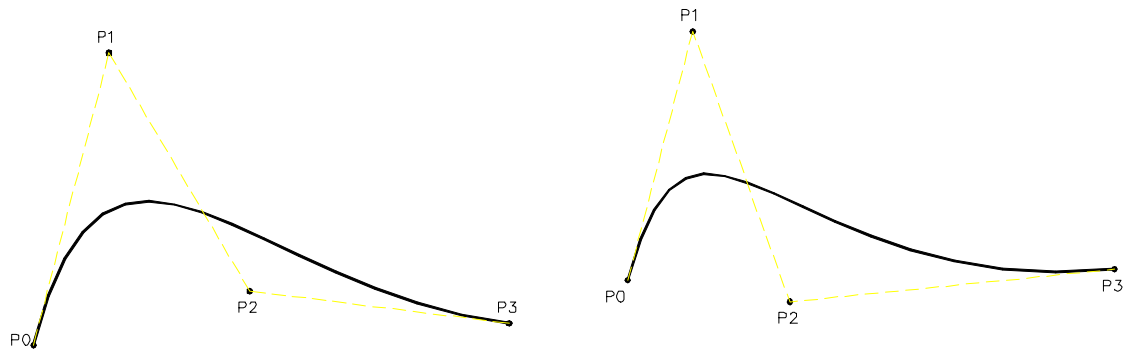


If the first vertex point is not programmed, the tangent at the end of the previous spline is used as the tangent at the start of the spline.

TANGENT AT END OF SPLINE

From the coordinates of the second vertex point (X62=, Y62=, Z62=) the tangent at the end of the spline is calculated as the line through the second vertex point and the end point.

CHANGING THE LOCATION OF THE SECOND VERTEX POINT



In the illustration three points of the spline are fixed and just the location of the second vertex point is changed. Notice that the shape of the curve is influenced by this change in location.

CONSTANT CURVATURE

If the first and second vertex points are not programmed, constant curvature between the splines is assumed.

GETTING STARTED WITH A GROUP OF BEZIER SPLINES

If a curve or surface is defined by a group of G6-blocks with Bezier splines, the first spline must be programmed with three vertex points.

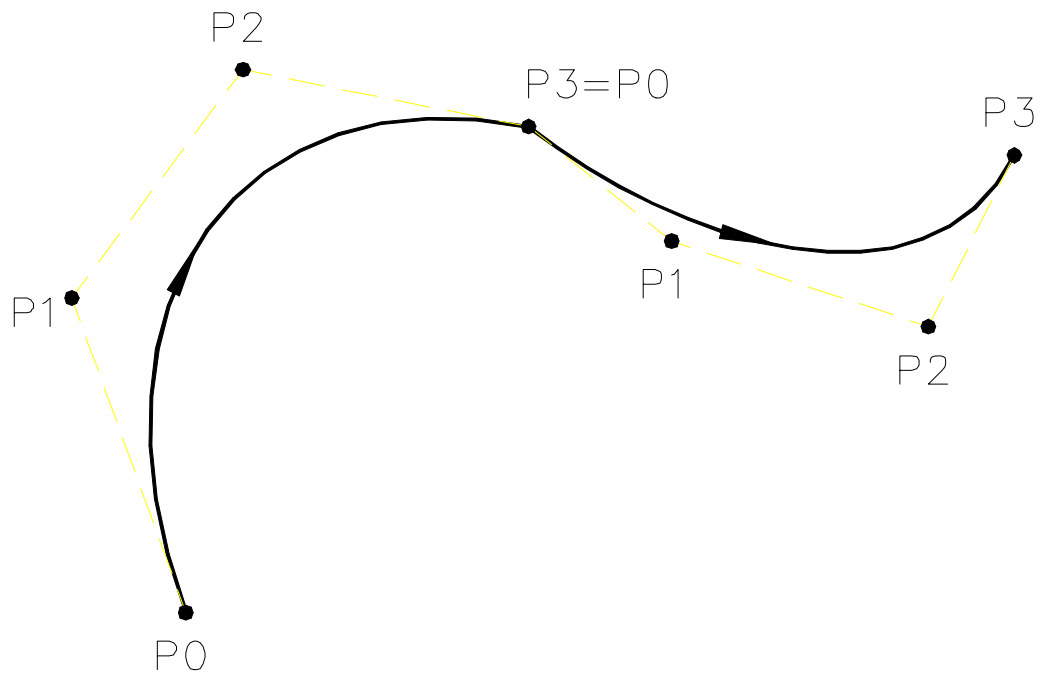
One way of doing this is to program a spline with three equal vertex points. These points are on a straight line, which is tangent to the spline in the end point of the "line".

Example: N10 G1 X10 Y10 Z10
 N11 G6 X61=25 Y61=25 Z61=10 X62=25 Y62=25 Z62=10 X25 Y25 Z10
 The block N11 generates a straight line in the XY-plane.

CONNECTING SPLINES

The three different types of Bezier splines can be connected to form a curve or surface.

CORNER BETWEEN CURVES



NB9770

If a corner between the splines, the second spline must be programmed with three vertex points.

RADIUS COMPENSATION

Radius compensation on splines is not available, so the toolpath must be programmed when using splines.

CONNECTION BETWEEN A LINE OR CIRCLE AND A BEZIER SPLINE

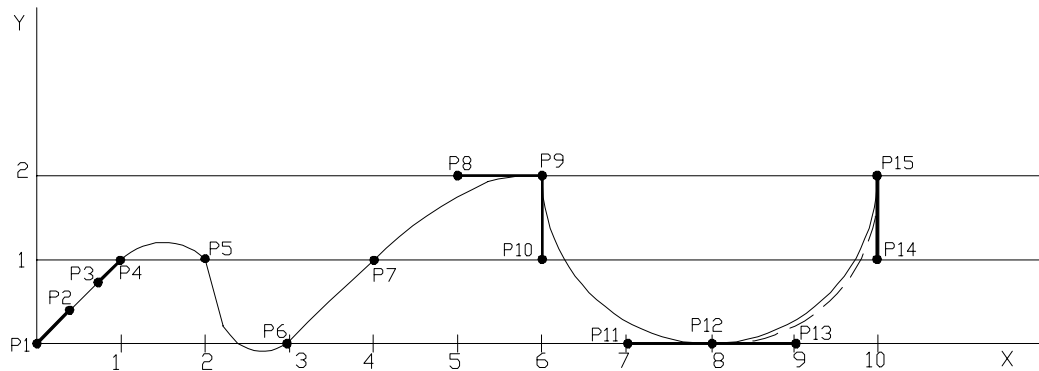
If the previous movement is linear or circular, its end point can be used as the start point of a Bezier spline with three vertex points.

Because radius compensation with splines is not available, the end point of the line or circle must also be programmed without radius compensation.

ABSOLUTE AND INCREMENTAL PROGRAMMING (G90/G91)

If G91 is active during a section with G6-blocks, the incremental programming is ignored with the G6-blocks and executed with the linear or circular movements.

It is advised to use always absolute programming (G90 active), when splines are involved.

Example:

```

N17001 (Spline curve)
N1 G98 X2 Y-6 Z-2 I10 J10 K10
N2 G17
N101 G0 X0 Y0 Z0 F500
N102 G6 X1 X61=0.3 X62=0.7 Y1 Y61=0.3 Y62=0.7 Z0.001 Z61=0 Z62=0
N103 X2 Y1.001 Z0
N104 X3 Y0 Z0.001
N105 X4 Y1 Z0
N106 X6 X62=5.7 Y2 Y62=2 Z0.001 Z62=0
N107 X8 X61=6 X62=7.5 Y0 Y61=1.5 Y62=0 Z0 Z61=0 Z62=0.001
N108 X10 X61=8.5 X62=10 Y2 Y61=0 Y62=1.5 Z0.001 Z61=0.001 Z62=0
N109 G0 X0 Y0 Z0
N110 M30

```

Explanation:

- N101: Approaching starting position of curve (P1)
- N102: First curve element. Straight line. Touches P1-P2 and P3-P4. End point is P4. All coordinates should be entered. Select a straight line.
- N103: Curve passes through P5
- N104: Curve passes through P6
- N105: Curve passes through P7
More points should be added if the curve differs from the required shape.
- N106: Curve passes through P9 and touches line P8-P9.
- N107: New curve with sharp transition is defined. First curve element starts in P9 and touches P9-P10 and P11-P12. End point is P12.
- N108: New curve with tangential transition is defined. First curve element starts in P12 and touches P12-P13 and P14-P15. End point is P15. The radius of curvature may be adjusted in P15 by changing distance P14-P15.
- N109: Return to starting position.

Note: In G6, the same coordinates should be different in two blocks (Z0 and Z0.001)

Notes and usage with cubic splines

In-depth knowledge of the cubic spline coefficients is essential. Calculations are usually done by a CAD system. The toolpath is then also generated.

POLYNOMIAL EXPRESSION FOR CUBIC SPLINE

The cubic spline is defined by a polynomial expression.

For e.g. the X-axis, the polynomial expression for the cubic spline is $(t \text{ is a parameter with } 0 < t < 1)$:

$$X = [X53 =]t^3 + [X52 =]t^2 + [X51 =]t$$

For the other axes the same polynomial is used with the programmed coefficients and the same parameter t for all axes.

These three polynomial expressions define a patch of the surface in space.

DETERMINING THE COEFFICIENTS

It is not easy to calculate the coefficients of the cubic polynomial. A good knowledge of spline coefficients is required. These calculations are mostly performed by a CAD-system, which also generates the toolpath.

TANGENTIAL CONTINUITY AT START OF SPLINE

If the first order coefficients are not programmed, the tangent at the end of the previous spline is used as the tangent at the start of the spline. From this tangent line the missing coefficients are calculated.

CONSTANT CURVATURE

If the first and second order coefficients are not programmed, constant curvature between the splines is assumed. The missing coefficients are calculated by the control.

Note: Omitting coefficients is only possible if the missing coefficients can be calculated from previous spline blocks

GETTING STARTED WITH A GROUP OF CUBIC SPLINES

If a curve or surface is defined by a group of G6-blocks with cubic splines, the first spline must be programmed with all three coefficients.

Notes and usage with both types of splines

MIXING BEZIER AND CUBIC SPLINES

Both types of splines can be mixed at will.

GRAPHIC SIMULATION

The splines can be displayed in synchron graphics. Other graphic modes ignore the spline function.

CANCELLATION

The function G6 is cancelled by one of the functions G0, G1, G2 or G3 at end of program (M30), CLEAR CONTROL or the softkey CANCEL PROGRAM.

PLANE SELECTION (G17/G18/G19)

Splines are independent of the selected plane. The points (coefficients) determine the plane in which the spline is made.

ZERO POINT SHIFT (G92/G93)

If a zero point shift is programmed between G6-blocks, it is ignored with the G6-blocks and used with the other blocks.

AXIS ROTATION (G92/G93 B4=..)

Axis rotation is ignored with the G6-blocks and used with the other blocks.

SCALING AND MIRROR IMAGE (G73)

Scaling and mirror image are ignored with the G6-blocks and used with the other blocks.

START NEXT MOVEMENT

In general feed movements are executed without a stop between the blocks. This results in rounded corners.

G28 and parameter I3= allows to program if the next movement starts after a full stop of the tool or without a stop between the movements.

Refer to the function G28 for details.

RESTRICTIONS

1. Corner accuracy (G28 I3=2 or 3) and feed limitations (G28 I6=) cannot be used together with spline functions
2. The BTR-function cannot be used with splines.
3. When geometric calculation is active (G64) the spline-interpolation cannot be used.

5.6 G7 Tilting working plane

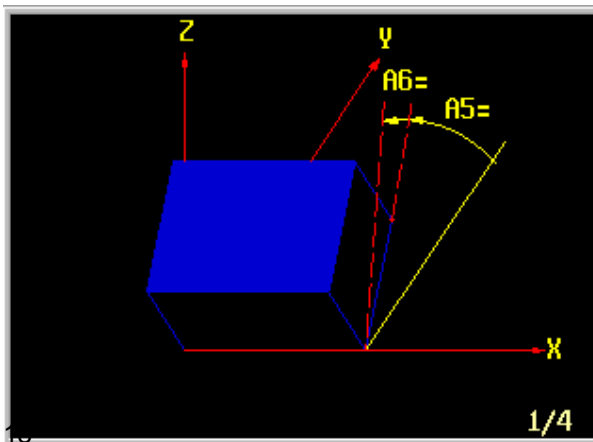
Purpose

Programming of a tilt-operating plane for four or five axis machines.

The position of the operating plane can be tilted using the function "Tilt operating plane". The operation, which has then been programmed in the principal plane, (G17, G18) can then be implemented within the tilt-operating plane. The tool axis is then orientated vertically in the new plane.

The tilt of the operating planes is defined and implemented using the G7 function.

Refer to chapter 14.1 "Tilting of the operating plane" on page 679



Format

N.. G7 {A5=.. | A6=..} {B5=.. | B6=..} {C5=.. | C6=..} {A7=..} {B7=..} {C7=..} {B47=..} {L1=..} {L..}

Parameter

```
G   Tilting working plane
L   Tool length offset
B47= E-par. for rotation mainplane
A5=  Angle of rotation absolute
B5=  Angle of rotation absolute
C5=  Angle of rotation absolute
A6=  Angle of rotation incremental
B6=  Angle of rotation incremental
C6=  Angle of rotation incremental
A7=  E-par. for position in A
B7=  E-par. for position in B
L1=  0=No move.,1=rot.axes,2=tooltip
```

18

Associated function

G-FUNCTIONS, WHICH ARE NOT PERMITTED, IF G7 IS SWITCHED ON

IF G7 is switched on, the following (modal) G-functions are not allowed to be active:

G6, G9, G19, G41, G42, G43, G44, G61, G64, G73, G141, G182, G197, G198, G199, G200, G201, G203, G204, G205, G206, G207, G208

When G7 is turned on, the following (modal) G-functions with the addresses below must not be active: G54 I1 B4=... and G93 B4=...

G FUNCTIONS WITHIN G7, WHICH ARE NOT PERMITTED

If G7 is active, the following G functions are not permitted:

G6, G19, G182

G FUNCTIONS, WHICH ARE NOT PERMITTED, IF G7 IS SWITCHED OFF

If G7 is switched off, the following (modal) G functions are not allowed to be active:

G9, G41, G42, G43, G44, G61, G64, G73, G141, G197, G198, G199, G200, G201, G203, G204, G205, G206, G207, G208

If one of these G functions which are not permitted is active, the fault report P77 "G function and Gxxx are not permitted" is generated.

When G7 is turned off, the following (modal) G-functions with the addresses below must not be active: G54 I1 B4=... and G93 B4=...

Type of function

Modal

Notes and usage

G7 FUNCTION

The freely programmable operating plane is defined using the new G7 function:

The new plane becomes active with the original null point.

The tool is orientated vertically in the new plane. The axes, which move depends upon the machine configuration and the programming.

The display shows the coordinates in the new (tilt) plane.

The manual operation is orientated in accordance with the new plane.

SPACE ANGLE

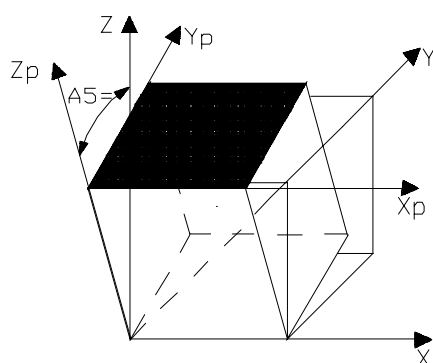
A5=, B5=, C5=

Defines the absolute angel, by which the operating plane is rotated about the corresponding positive axis.

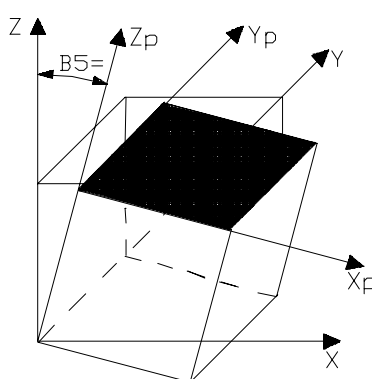
A6=, B6=, C6=

Defines the incremental angle, by which the operating plane is rotated about the corresponding positive axis.

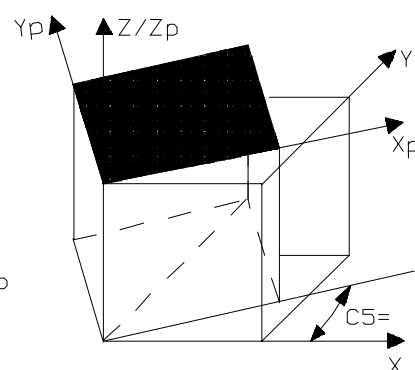
Value falls between -359.999 and 359.999 [degrees]



N... G7 A5=



N... G7 B5=



N... G7 C5=

DEFINITION OF NEW OPERATING PLANES

Tilting of the operational plane can be defined in two ways:

- Programming with A5=, B5= or C5= parameters. In this way, the absolute tilts about the corresponding positive axes are defined. The tilts are implemented as follows:
 1. the active G7 tilt is raised
 2. C5= tilt about the machine fixed positive Z axis
 3. B5= tilt about the positive Y axis
 4. A5= tilt about the positive X axis
- Programming with A6=, B6= or C6= parameters. The incremental tilts about the current corresponding positive axes are defined in this way. The tilts are implemented as follows:
 1. C6= tilt about the current G7 positive Z axis
 2. B6= tilt about the current G7 positive Y axis
 3. A6= tilt about the current G7 positive X axis

The programming is independent of the machine configuration. The plane tilt is calculated with reference to the current null point. The movement is dependent upon the machine configuration.

QUERY A CALCULATED ANGLE POSITION

- | | |
|---------------|--|
| A7=, B7=, C7= | Holds the number of the E-Parameters, in which the computed angle of the corresponding rotary axis is set. |
| B47= | Contains the number of the E-Parameter, in which the computed angle of the main plane is set. |

TOOL VERTICAL ON THE DEFINED TILT PLANE

The G7 tilt movement takes place interpolating with the power traverse. It tilts the tool axis to the defined plane. The axes, which are moved, depend upon the type of movement L1=:

- L1=0 the rotary axes do not move (start position).

Comment:

The tilt movement can then be implemented, using the E parameters loaded into A7=, B7= or C7=. This movement must then be programmed manually.

- L1=1 Interpolate only the rotary axes, which do not move the linear axes.
- L1=2 Interpolate the rotary axes and to that end execute a "compensatory movement". In this way the tooltip remains in the same position with respect to the workpiece.

TOOL LENGTH ALLOWANCE

If the tilting motion takes place about the tooltip (L1=2), I defines an allowance in the tool direction between the programmed endpoint and the tooltip.

SWITCHING OFF THE G7 FUNCTION

The operation of G7 remains active until G7 is switched off. G7 is switched off by the programming of G7 without parameters or by G7 L1=1 positioning of the rotary axes on the workpiece null point.

G7 is not switched off by M30 or <Program interrupt>. After switching on the control G7 is permanently active. Travel in the G7 plane is then possible. G7 is switched off in accordance with reference point travel or <Reset CNC>.

Note: It is recommended that, at the start of every G7 program, that a G7 without parameters is programmed. In this way, during the start-up of the program (interrupt within the tilted plane and the new start), the plane is always reset. Without this G7 at the start, the first part of the program will be implemented in the tilted plane rather than in the untilted plane. This programming is similar to programming with G17/G18 - different null points or different tools.

ROTARY AXES

Rotary axes can be programmed in the tilted planes in the normal way. It is the programmer's responsibility to ensure compatibility of the rotary axes with the G7 tilt.

ABSOLUTE POSITION G74

If G7 is active, the "Absolute position" G74 is referred to the machine coordinates. This is the same as in V3.3x.

GRAPHICS

The graphics display the G7 plane as the main picture. The screen is refreshed whenever G7 becomes active.

If G7 is active, the position between tool and workpiece is displayed.

DISPLAY

If G7 is active, a yellow icon is displayed on the screen behind the tool number. By means of a small "p" on the right next to the "axes characters", an indication is given as to whether the display relates to the tilted operating planes or to the machine coordinates. The operating status has been enhanced with the current reading of the programmed G7 space angle.

A new soft key (Jog to the G7 plane) appears in the "Jog operation type" soft key group. This soft key is used to switch between the tilted operating planes and the machine coordinates. If the position is displayed in machine coordinates, the actual position of the tooltip is shown.

CHANGE OF TOOL

If G7 is active, a tool change is not permitted (fault report). G7 must first be deselected. G7 must then be selected again, in order to resume work in the tilted plane following the tool change.

Example:

```
N100 G7 B5=45 L1=1    (plane is set)
N110 T14              (tool preselect)
..
N200 G0 Z200          (the tool axis is withdrawn)
N210 G7 B5=0 L1=1    (deselect G7)
N220 M6               (tool change)
N230 G0 X.. Y.. Z..   (power traverse to the new start position)
N240 G7 L1=1 B5=45    (face is rotated again in the G7 plane)
```

PALETTE, TILT FACE OR TOOL CHANGE

While G7 is active a change of palette, tilt face or tool cannot be implemented. A fault is displayed and the program must be interrupted. Before such a change, G7 must be deactivated.

TILT OPERATING PLANE WITH M53/M54

During mixed operation with G7 and M53/M54, the tilt face positioning M53/M54 with M55 must be relinquished before programming G7. In this way, the face offset, which is active under these conditions, is relinquished.

M FUNCTIONS, WHICH ARE NOT PERMITTED IF G7 IS SWITCHED ON

If G7 is switched on, the following M functions are not permitted to be active:

M53, M54

M FUNCTIONS, WHICH ARE NOT PERMITTED WITHIN G7

If G7 is active, the following M functions are not permitted:

M6, M46, M53, M54, M60, M61, M62, M63, M66

FAULT REPORTS

P77 G-function and Gxxx not permitted

This error message tells, which combination of G-functions is not permitted. For example: Is G7 programmed and G41 is active, an error message P77 "G-function and G41 not permitted" is given.

P306 Plane not clearly defined

The G7 plane is defined with a mix of absolute (A5=, B5=, C5=) and incremental (A6=, B6=, C6=) angles.

Resolution: Use only absolute or incremental angles. If required, several G7 definitions with incremental angles behind one another can be defined.

P307 Programmed plane not attainable

The defined G7 tilt position, owing to the restricted range of the rotary axis, cannot be attained.

MACHINE CONSTANTS

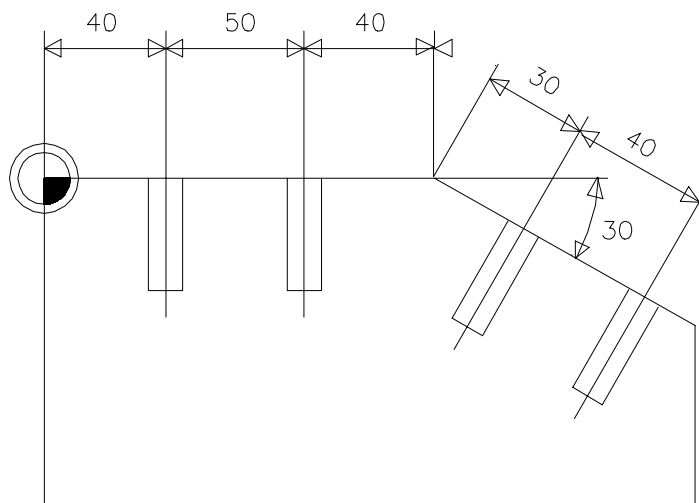
MC312 free operating plane (0=off, 1=on)

Activates the free operating plane. The G7 function can be programmed.

MC755 free operating plane: rotation (0=coordinates cross, 1=axes)

If the desired rotation of the operating plane is compatible with the rotation of a rotary axis, an adjustment may be made to determine whether the relevant rotary axis or the coordinates cross is rotated.

e.g. on a machine with (real C axis) the programming G7 C5=30 and MC755=0 generates a rotation of the coordinates cross through -30° and, if MC755=1, a rotation of the C axis through 30° .

Example 1 Workpiece with tilted operating plane.

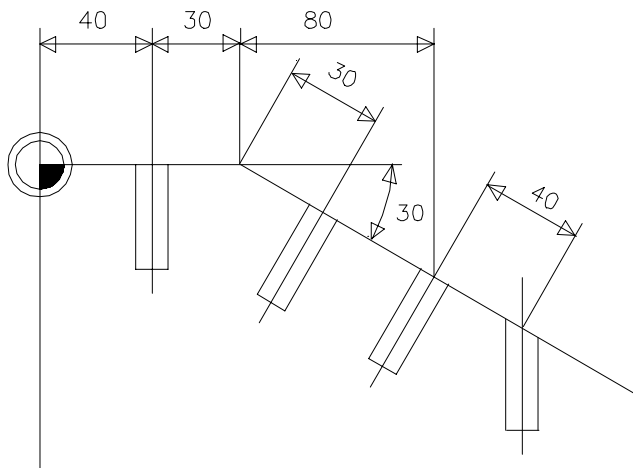
```

N10 G17
N20 G54
N30 M55
N40 G7 L1=1
N..
N100 G81 Y1 Z-30
N110 G79 X40 Z0
N120 G79 X90
N..
N200 G0 X130 Z50
N210 G93 X130
N220 G7 B5=30 L1=2 L50
N230 G79 X30 Z0
N240 G79 X70
N..
N300 G7 L1=2 L50

```

Explanation:

N10	Define operating plane
N20	Zero point shift
N30	Deselect ion M53/M54
N40	Reset G7
N100	Drill cycle definition
N110	Drill the first hole in the horizontal plane
N120	Drill the second hole in the horizontal plane
N..	Other movements in the horizontal plane
N200	Tool is set at the safety distance
N210	Zero point is set at the start of the tilted operating plane
N220	G7 Define new operating plane
B5=30	Angle of rotation
L1=2	Tool/table is rotated about the tooltip
L50	Extra length oversize in the direction of the tool. In this way, the tool is rotated about the zero point. The distance from the tooltip to the zero point is 50 mm
N230	Drill the first hole in the tilted operating plane
N240	Drill the second hole in the tilted operating plane
N..	Other movements in the tilted operating plane
N300	Reverse rotation in the horizontal plane

Example 2 Workpiece using tilted operating plane.

```

N10 G17
N20 G54
N30 M55
N40 G7 L1=1
N..
N100 T1 M6
N110 G81 Y1 Z-30
N120 G79 X40 Z0
N..
N200 T2 M6
N210 X70 Z50
N220 G93 X70
N230 G7 B5=30 L1=2 L50
N240 G1 X0 Z0
N250 X150
N..
N300 T1 M6
N310 G79 X30 Z0
N320 G92 X=80:cos(30)
N330 G79 X0 Z0
N..
N400 G92 X=40
N410 G0 X0 Z50
N420 G7 B5=0 L1=2 L50
N430 G79 X0 Z0
N..
N500 M30

```

Explanation:

N10	Defining operating plane
N20	Zero point shift
N30	Deselect ion of M53/M54
N40	Reset G7
N100	Tool change (drill)
N110	Define the drilling cycle
N120	Drilling of a hole in the horizontal plane

- N.. Other movements in the horizontal plane
- N200 Tool change (mill)
- N210 Tool is set at the safety distance
- N220 Define the zero point of the new operating plane
- N230 G7 Define new operating plane
 B5=30 Angle of rotation
 L1=2 Tool/table is rotated about the tooltip
 L50 Extra length oversize in direction of the tool. In this way the tool is rotated about the zero point. The distance of the tooltip from the zero point is 50 mm.
- N240 Positioning of the mill perpendicular to the tilted plane
- N250 Mill in the tilted plane
- N.. Other movements in the tilted operating plane
- N300 Tool change (drill)
- N310 Drilling the first hole in the tilted operating plane
- N320 Incremental zero point shift
- N330 Drilling the second hole in the tilted operating plane
- N.. Other movements in the tilted operating plane
- N400 Incremental zero point shift
- N410 Tool is set at the safety distance
- N420 G7 Deselect "Tilt operating plane"
 Reverse rotation in the horizontal plane
 B5=30 angle of rotation
 L1=2 Tool/table is rotated about the tooltip
 L50 Extra length oversize in the direction of the tool. In this way, the tool is rotated about the zero point. The distance of the tooltip from the zero point is 50 mm
- N430 Drilling the third hole in the horizontal operating plane
- N.. Other movements in the horizontal operating plane
- N500 Program end.

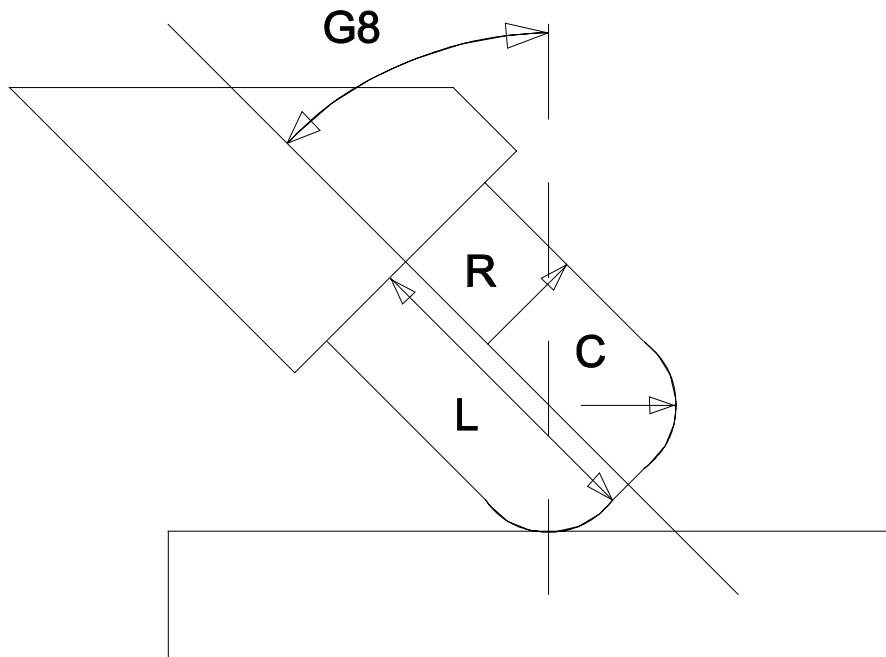
5.7 G8 Tilting tool orientation

Purpose

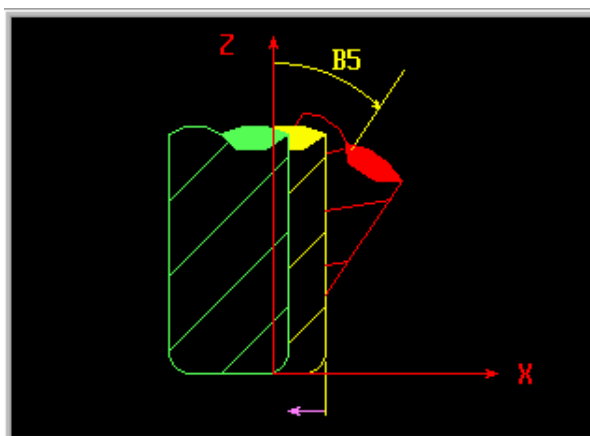
To program a swivelled tool for four or five-axis machines.

With the function "Swivel tool" the tool axis can be set obliquely relative to the machining plane. This makes angle milling possible and substantially improves cutting conditions and thus surface quality.

The programming of G8 is identical to G7. G7 should therefore also be read.



L, R and C from the tool table.



Format

N.. G8 {A5=.. | A6=..} {B5=.. | B6=..} {C5=.. | C6=..} {A7=..} {B7=..} {C7=..} {L} {L1=..} {F6=..} {F}

Parameters

```

G   Tilting tool orientation
L   Tool length offset
A5= Angle of rotation absolute
B5= Angle of rotation absolute
C5= Angle of rotation absolute
A6= Angle of rotation incremental
B6= Angle of rotation incremental
C6= Angle of rotation incremental
A7= E-par. for position in A
B7= E-par. for position in B
L1= 0=No move.,1=rot.axes,2=tooltip
L3= Radius compensation (0=on, 1=off)
F6= Block feed

```

Associated function**G FUNCTIONS NOT PERMITTED WITHIN G8**

The following G functions are not permitted when G8 is active:

G6, G19, G40, G41, G42, G43, G44, G141, G180, G182

Type of function

modal

Notes and use**SOLID ANGLE**

A5=, B5=, C5= Defines the absolute angle by which the tool direction rotates relative to the 'normal' tool axis.

A6=, B6=, C6= Defines the incremental angle by which the tool direction rotates relative to the 'normal' tool axis.

Value between -359.999 and 359.999 [degrees]

REDEFINE TOOL DIRECTION

The rotation of the tool direction can be defined in two ways:

- Programming with A5=, B5= or C5= parameters. This defines the absolute rotations about the corresponding positive axes. The rotations are calculated as follows:
 1. the active G8 rotation is cancelled
 2. C5= rotation about the positive Z axis fixed with respect to the machine
 3. B5= rotation about the positive Y axis
 4. A5= rotation about the positive X axis
- Programming with A6=, B6= or C6= parameters. This defines the incremental rotations about the corresponding current positive axes. The rotations are calculated as follows:
 1. C6= rotation about the current G8 positive Z axis
 2. B6= rotation about the current G8 positive Y axis
 3. A6= rotation about the current G8 positive X axis

Programming is independent of the machine configuration. The plane rotation is calculated relative to the current zero point. The motion is dependent on the machine configuration.

SCANNING A CALCULATED ANGULAR POSITION

A7=, B7=, C7= Contains the number of the E parameter in which the calculated angle of the corresponding rotary axis is set.

FEED

F6= is a local feed which is only active in the record in which it is programmed. In this case, it is the rotation of the tool. F is the normal feed and also applies to the subsequent records.

SWIVEL MOTION

The G8 swivel motion is performed by interpolating with feed (F6=). It swivels the tool axis onto the defined plane. Which axes move depends on the type of motion L1= :

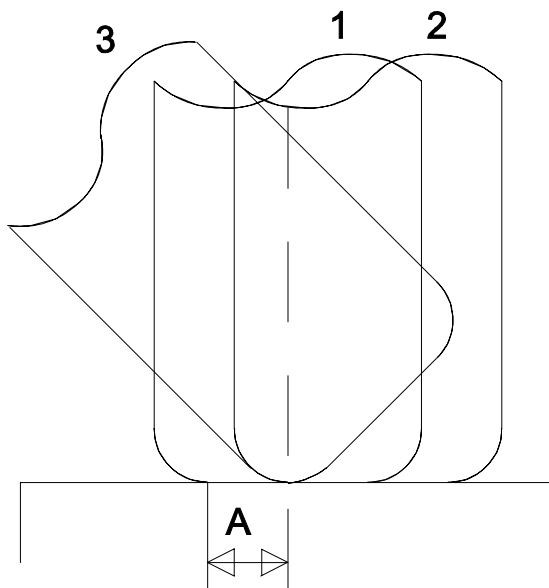
- L1=0 The rotary axes do not move (default).

Note: The swivel motion can be programmed or performed manually by means of the E parameters that are loaded with A7=, B7= or C7=.

- L1=1 Only the rotary axes swivel, while the linear axes do not move.

- L1=2 The rotary axes swivel and the linear axes perform a movement. This means that the contact point position remains X, Y, Z.

If the contact point lies on the tool corner radius, the movement is only a rotation. If the contact point is the tooltip and the corner radius (C) is smaller than the tool radius (R), a compensating movement occurs so that the contact point is shifted from the tooltip to the corner radius.



With cylindrical cutters (with corner radius $C < \text{cutter radius } R$), the following particular point applies:

When swivelling from the vertical (1) to the oblique (2--> 3) position or vice versa, the contact point moves from the centre of the cutter to the corner radius (A) and vice versa. A compensating movement at the tooltip ensures that the current contact position X, Y, Z remains unchanged.

CAUTION: The movements when applying/cancelling the tool correction within G8 may result in a collision. It is the responsibility of the programmer (operator) to avoid this.

TOOL LENGTH ALLOWANCE

If the swivel motion takes place about the tool contact point (L1=2), L defines an extra allowance in the tool direction between the rotation point and the tooltip.

TOOLRADIUS-CORRECTION (L3=)

Dependent of the toolradius-correction (L3=) the radiuscorrection will be calculated.

When L3=1 the radius (R) and the cornerradius (C) will be not considered. The tool turns around the tooltip and a compensation movement will not be made.

When L3=0 the radius (R) and cornerradius (C) will be considered. The tool turns and a compensation movement will be made.

Default L3=0.

TOOL CORRECTION

During the function "swivel tool" (G8) the values L, R and C, dependent of the toolradius-correction (L3=), for the tool are corrected.

This G8 tool correction is independent of G40, G41, G42, G43, G44 and is always effective.

At the beginning and end of the tool correction, a compensation movement is carried out if the corner radius (C) is smaller than the tool radius (R).

If the tool dimensions (L, R, C) change with G8 active, the current position of the linear axes is re-calculated.

TURNING OFF THE G8 FUNCTION

G8 remains active until it is cancelled. G8 is cancelled by programming G8 without angle parameters.

G8 is not cancelled by M30 or <program abort>. After the control is turned on, G8 is still active. After search for reference points or <CNC reset> G8 is cancelled.

Note: At the start any program that uses G8, we recommend that a G8 be programmed with no parameters. This ensures that the tool direction is always reset as the program is starting up (abort with swivelled tool and new start). Without this G8 at the beginning, the first part of the program is executed in the swivelled instead of the unswivelled plane.

The programming is similar to programming with G7/G17/G18 - different zero points or different tools.

CONFIGURATION

Swivel tool (G8) can be used for machines where a kinematic model is defined and entered. See description of the kinematic model.

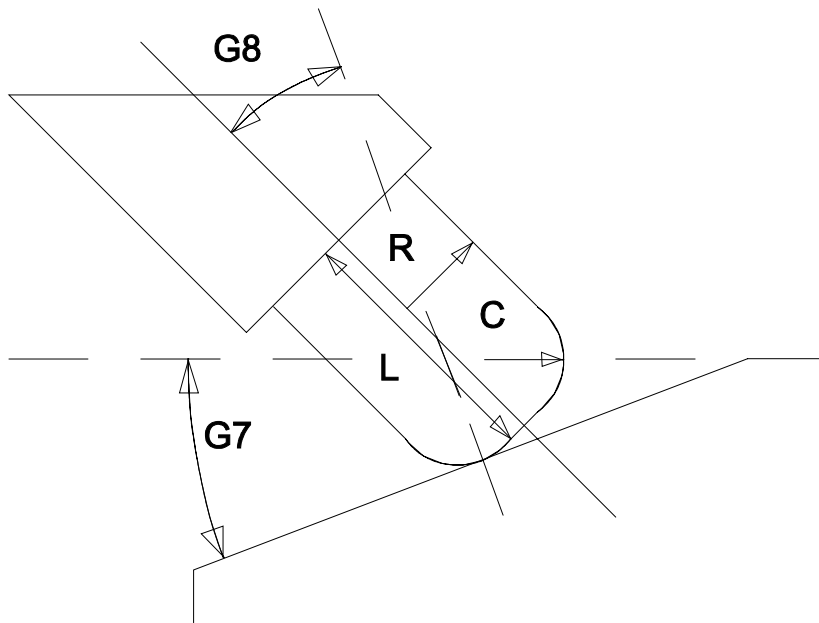
GRAPHICS

The G8 has no effect on the graphics.

DISPLAY

When G8 is active, a yellow icon is displayed in the display behind the tool number.

A small 'p' on the right, next to the 'axis letters', is used to display whether the position of the tooltip is displayed or the position in machine coordinates.

Example 1 Workpiece with oblique machining plane and oblique tool.

```

N10 G17
N20 G54
N30 M55
N40 G7 L1=1
N50 G8 L1=1
..
N100 G0 X130 Z50
N110 G93 X130
N120 G7 B5=-30 L1=2
N130 G8 B5=30 L1=2
..
N200 G8
N210 G7 L1=2

```

Key:

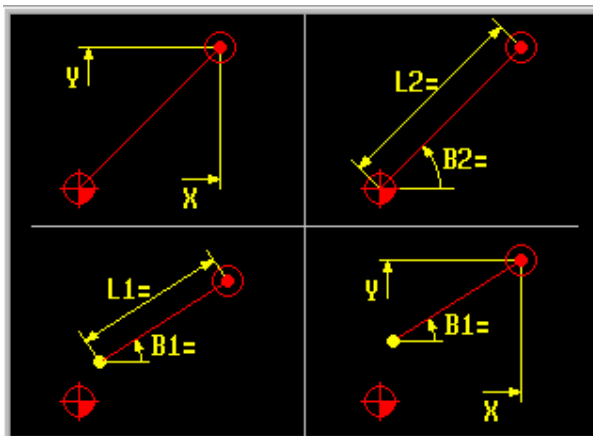
N10	Define machining plane
N20	Zero point offset
N30	Deselect M53/M54
N40	Reset G7
N50	Reset G8
N100	Tool set to safety distance
N110	Zero point set to the beginning of the swivelled machining plane.
N120	G7 Define new oblique position of the tool. B5=-30 Angle of rotation L1=2 Tool/table revolves about the tooltip
N130	G8 Define new oblique position of the tool. B5=30 Angle of rotation L1=2 Tool rotates about the tooltip and a compensation movement is performed.
N200	Turn tool perpendicular to the machining plane again (rotary and compensation movement).
N210	Rotate back to the horizontal plane.

5.8 G9 Define pole position (size reference point)

Purpose

To program a pole point. When a pole point has been programmed, program blocks with pole programming (angle and length) no longer relate to the zero point, but to the pole point most recently programmed.

The pole point is programmed as a function of the modally valid system of measurement G90/G91. Furthermore, the pole point may be programmed wordwise absolute, incrementally or combined absolute/incrementally.



Format

G17 active: N.. G9 X.. Y.. {X90=...} {X91=...} {Y90=...} {Y91=...}
 G18 active: N.. G9 X.. Z.. {X90=...} {X91=...} {Z90=...} {Z91=...}
 G19 active: N.. G9 Y.. Z.. {Y90=...} {Y91=...} {Z90=...} {Z91=...}

Deactivate pole (identical with workpiece zero point)
 N.. G9 X0 Y0

Pole point in polar coordinates (G17, G18, G19 active):
 absolute:
 N.. G9 B2=.. L2=.. {B1=..} {L1=..}

Parameter

```
G   Define pole position
X   Pole coordinate
Y   Pole coordinate
Z   Pole coordinate
B1= Angle
B2= Polar angle
?90= Pole coordinate abs. (X,Y,Z..)
?91= Pole coordinate incr. (X,Y,Z..)
L1= Path length
L2= Polar length
```

Associated functions

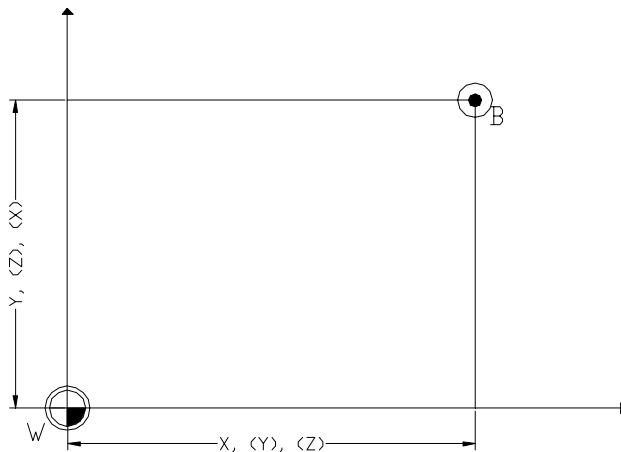
G0, G1, G2, G3, G11, G40, G41, G42, G43, G61, G62, G64, G77, G78, G79, G92, G93, G145,

Type of function

Modal

Notes and usage**Pole point in Cartesian coordinates:****Pole point in absolute coordinates:**

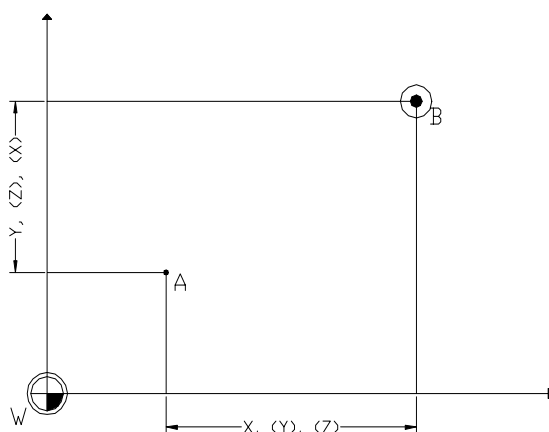
The programmed coordinates relate to the workpiece zero point.



B = Polpunkt
N.. G9 X.. Y..

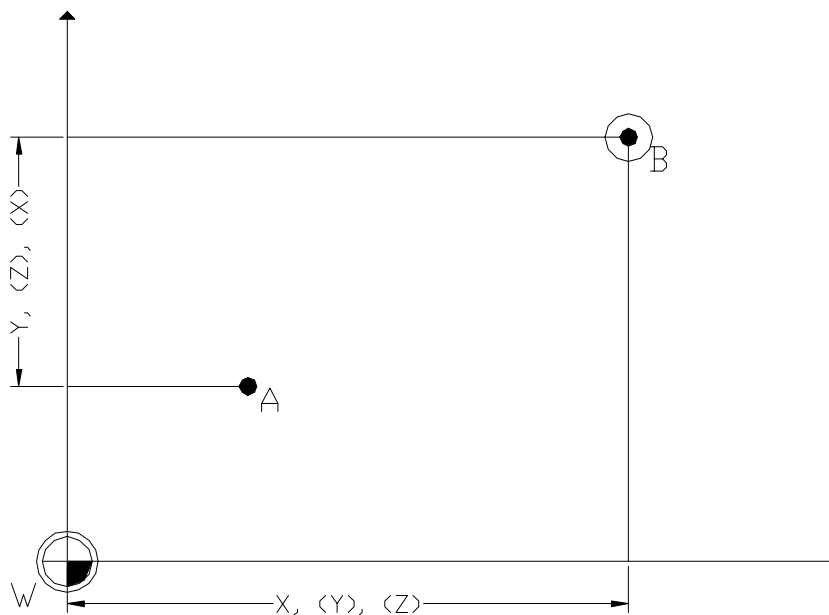
Pole point in incremental coordinates:

The programmed coordinates relate to the actual position



A = existing pole point
B = new pole point
N... G9 X91=... Y91=...

Pole point in combined absolute/incremental coordinates:

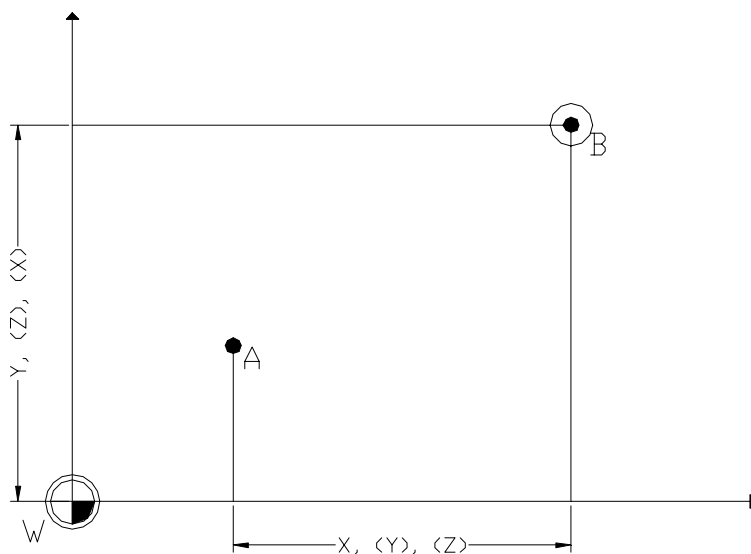


A = existing pole point

B = new pole point

N... G9 X... Y91=...

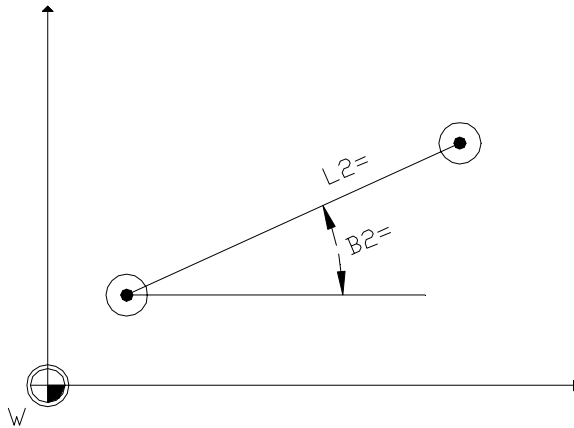
N.. G9 X91=.. Y..



N.. G9 X91=.. Y..

Pole point in polar coordinates (G17, G18, G19 active):**Pole point in absolute polar coordinates:**

The polar coordinates B2= and L2= relate to the most recently active pole point.



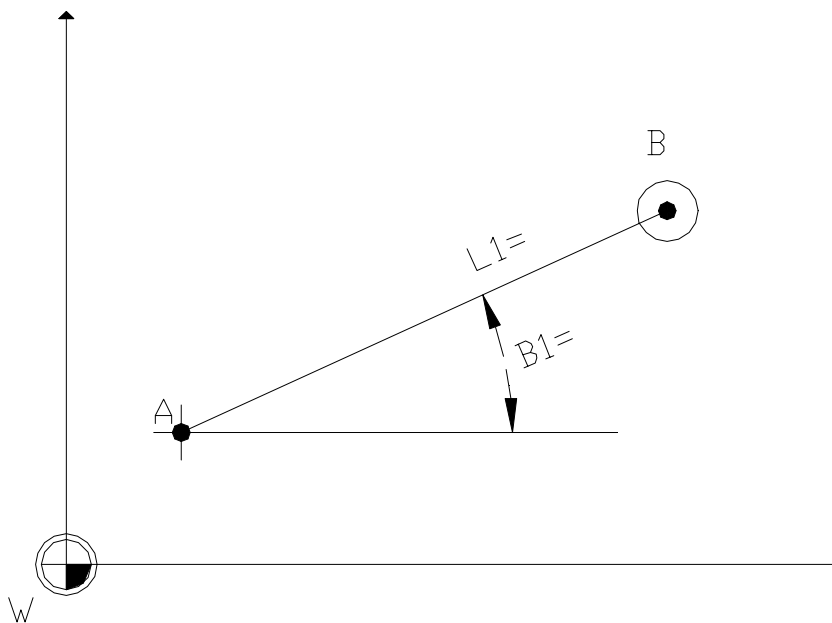
A = existing pole point

B = new pole point

N.. G9 B2=.. L2=..

Pole point in incremental polar coordinates:

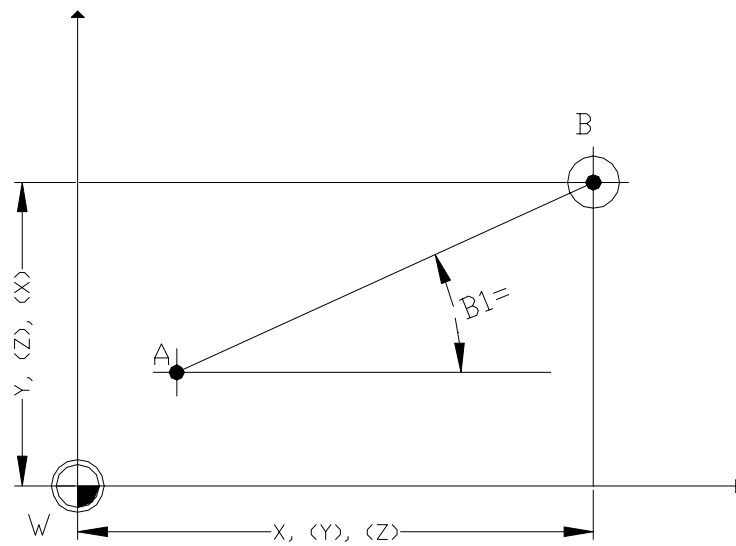
The polar coordinates B1= and L1= relate to the actual position.



A = end point of last movement

B = new pole point

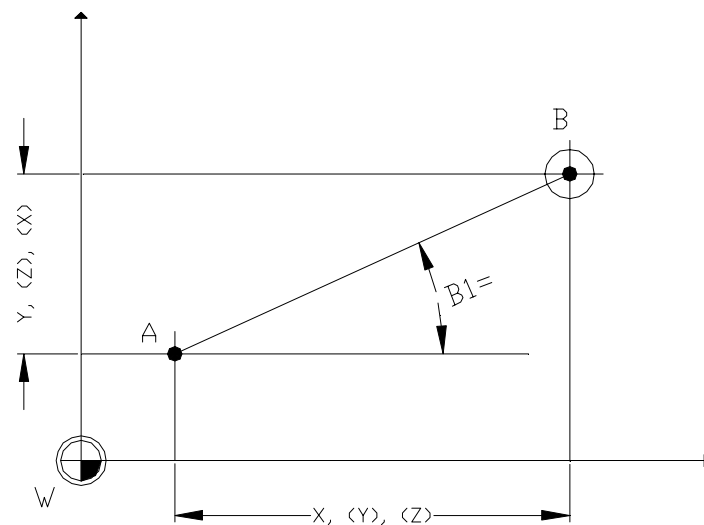
N.. G9 B1=.. L1=..

Combined programming: Cartesian absolute/polar:

A = existing pole point

B = new pole point

N.. G9 X.. B1=..

Combined programming: Cartesian absolute/polar:

A = existing pole point

B = new pole point

N.. G9 X91=.. B1=..

- polar definitions are allowed in the active working plane only
- before the G9 block is called, the pole point is at the workpiece zero point (pole point = 0)
- the pole point is modally active
- the pole point may be redefined indefinitely
- the pole point is zeroed (0) when changing the plane using G17, G18, and G19

Polar definition of end point:

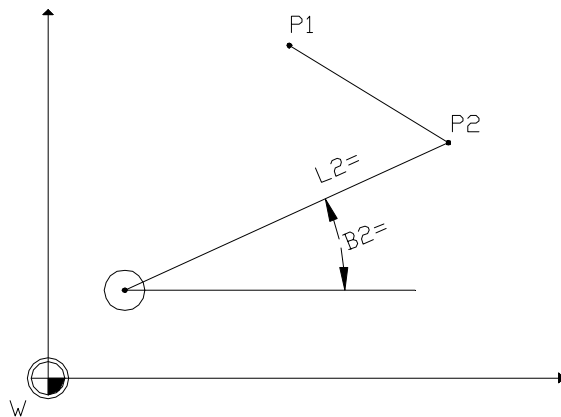
During absolute polar programming the polar lengths $L2=$ and $L3=$ and polar angles $B2=$ and $B3=$ no longer relate to the zero point, but to the pole point.

If no pole point has been defined, the pole point = 0 (zero) and therefore equals the active zero point.

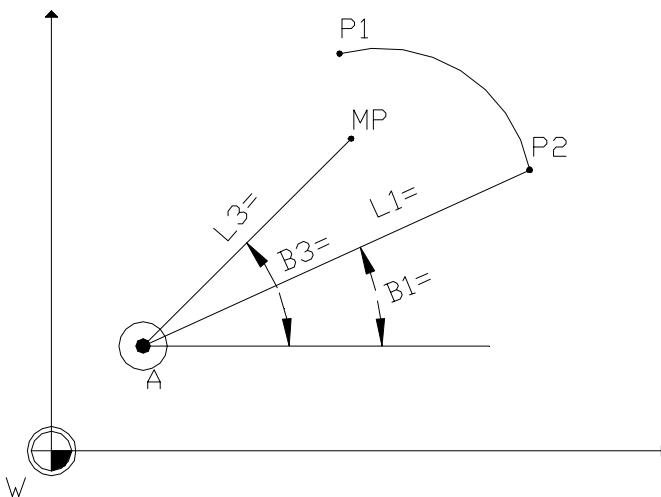
Pole point definition

Pole points with pole can be defined in the following G-functions:

G0, G1, G40, G44, G61, G62, G77, G78, G79, G145

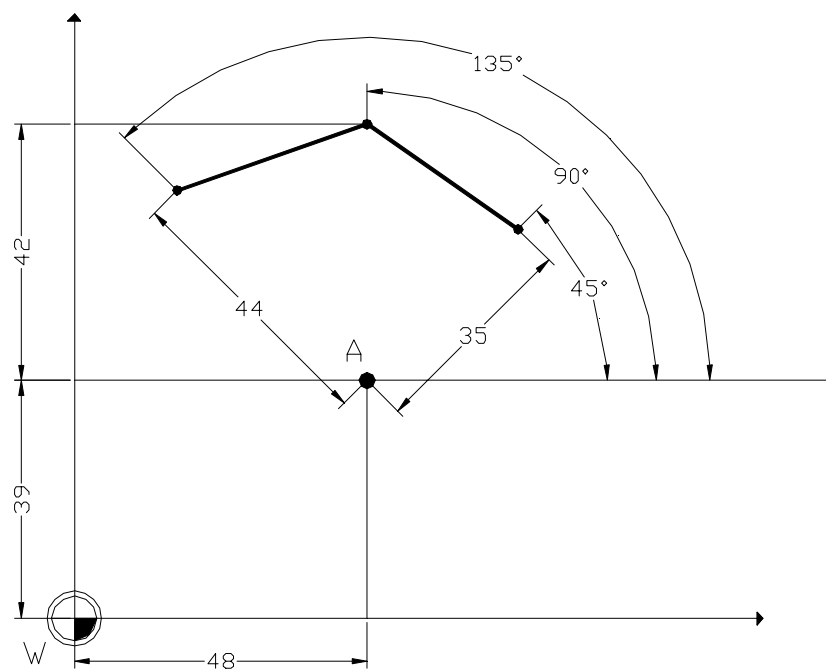
**Polar circle definition**

Polar programming with pole point of the centre point and end point is possible in G2 and G3 blocks.

**ICP/Geometry calculation G64**

G1, G2 and G3 blocks with $B2=$, $B3=$ and $L3=$ programming can be programmed in G64 and ICP. These blocks relate to the active pole point. The pole point can only be changed in G64 and not in ICP.

Example



B = new pole point

```

N30 G9 X48 Y39
N40 G1 B2=135 L2=44
N50 G1 B2=90 L2=42
N60 G1 B2=45 L2=35

```

Explanation:

N30 Definition of new pole point

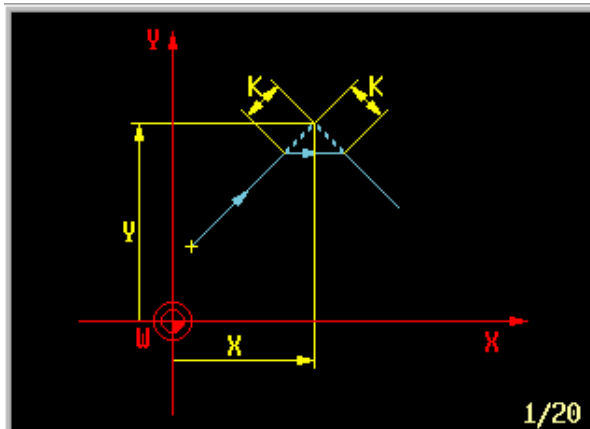
N40 Definition of end point coordinates related to new pole point

5.9 G11 One/Two point or two line geometry with chamfer or rounding

Note

Use of this function is limited only to programs made on earlier control systems.
The operator can easily make programs requiring geometry calculations, using Interactive Contour Programming (ICP).

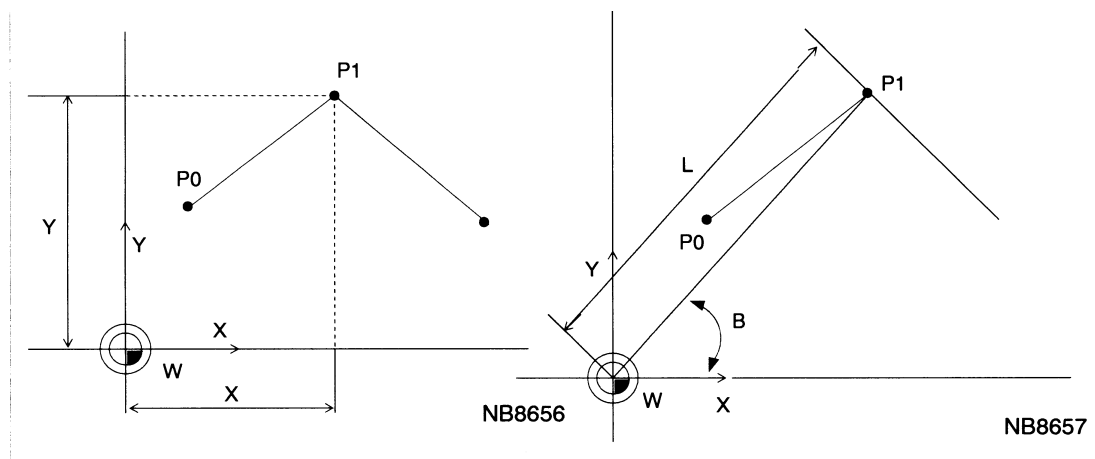
Purpose



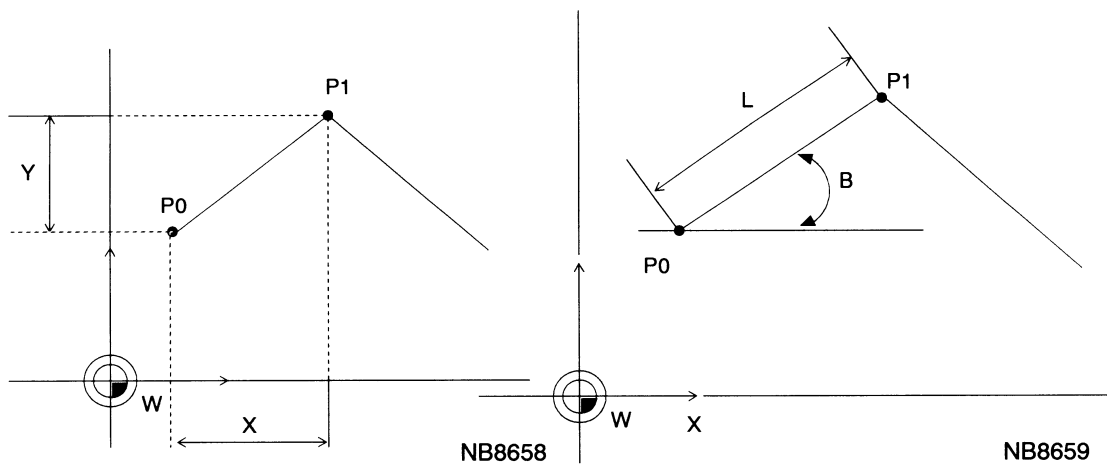
1. One Point Geometry

To program in one block:

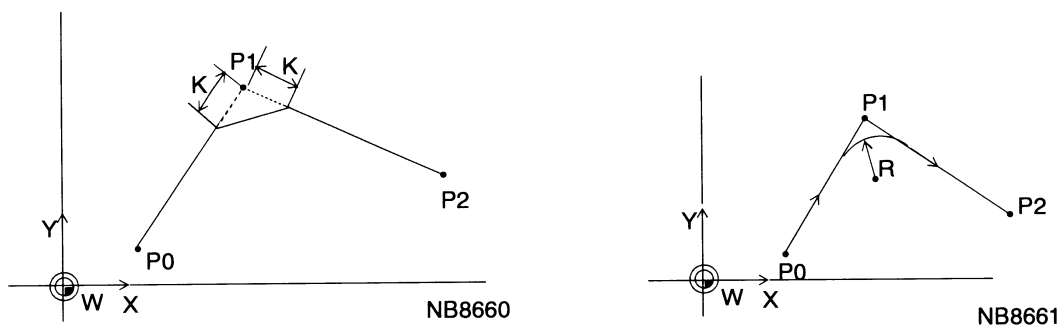
- The end point of a linear movement
- If required, a symmetrical chamfer or rounding between this movement and the next linear movement.



Absolute coordinates (G90)



Incremental coordinates (G91)

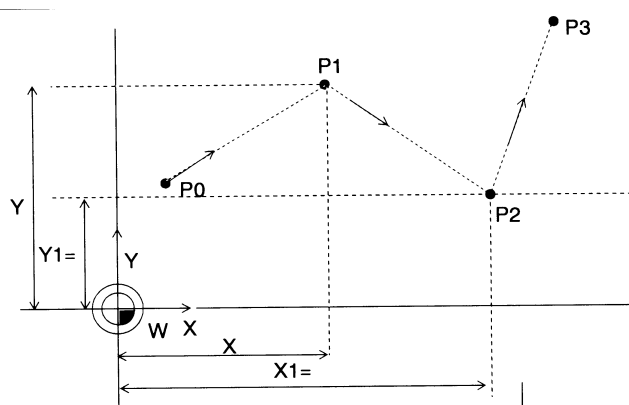


Chamfer rounding with one point geometry

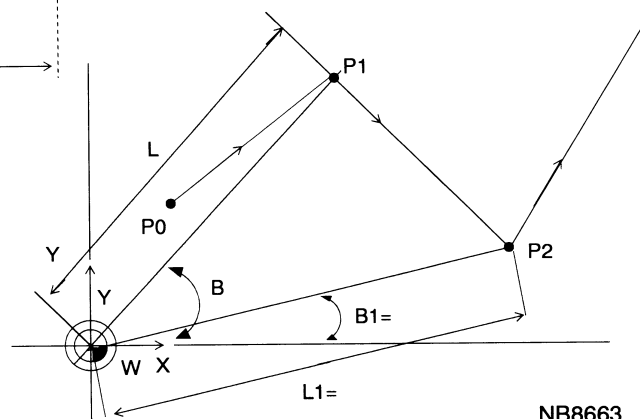
2. Two Point Geometry

To program in one block:

- The end points of two separate linear movements
 - If required, a symmetrical chamfer or rounding between these movements
 - If required, a symmetrical chamfer or rounding between the last movement and the next linear movement.
- Chamfers or roundings with two-point geometry.

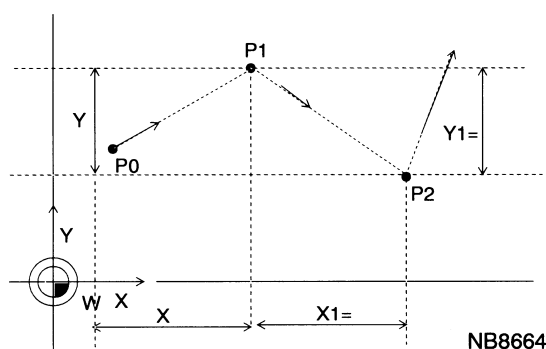


NB8662

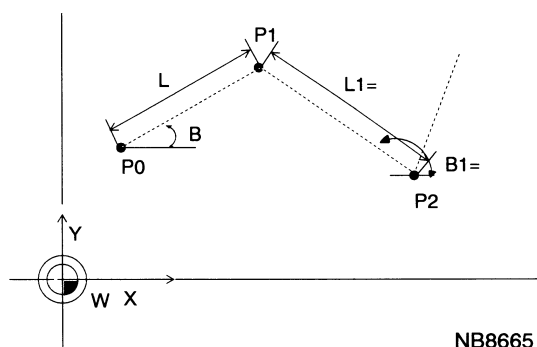


NB8663

Absolute coordinates (G90)

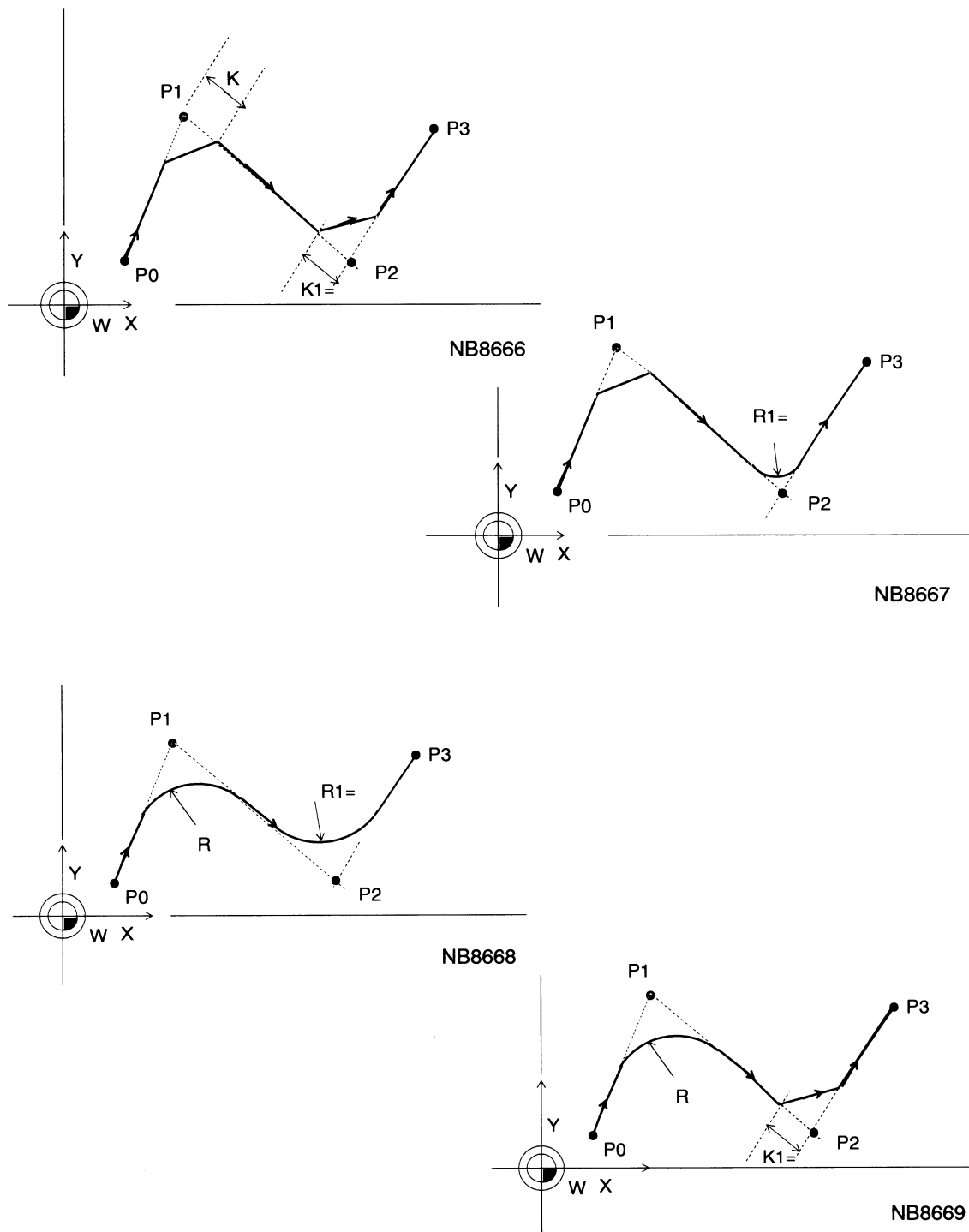


NB8664



NB8665

Incremental coordinates (G91)

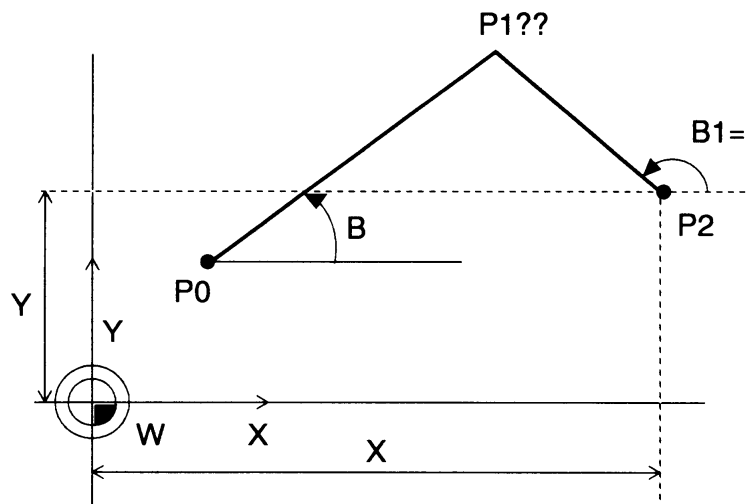


Chamfers or roundings with two-point geometry

3. Two Line Geometry

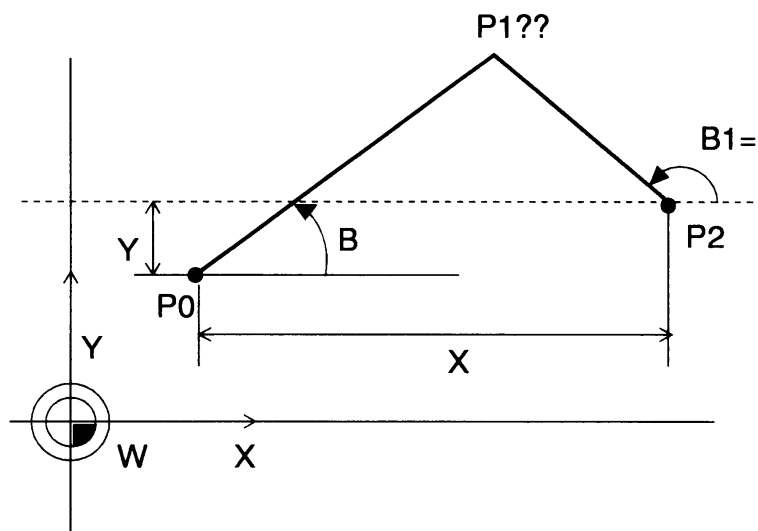
To program in one block two separate linear movements:

- The first linear movement with the angle with the main axis
- The second linear movement with the end point and the angle with the main axis
- If required, a symmetrical chamfer or rounding between these movements
- If required, a symmetrical chamfer or rounding between the last movement and the next linear movement.



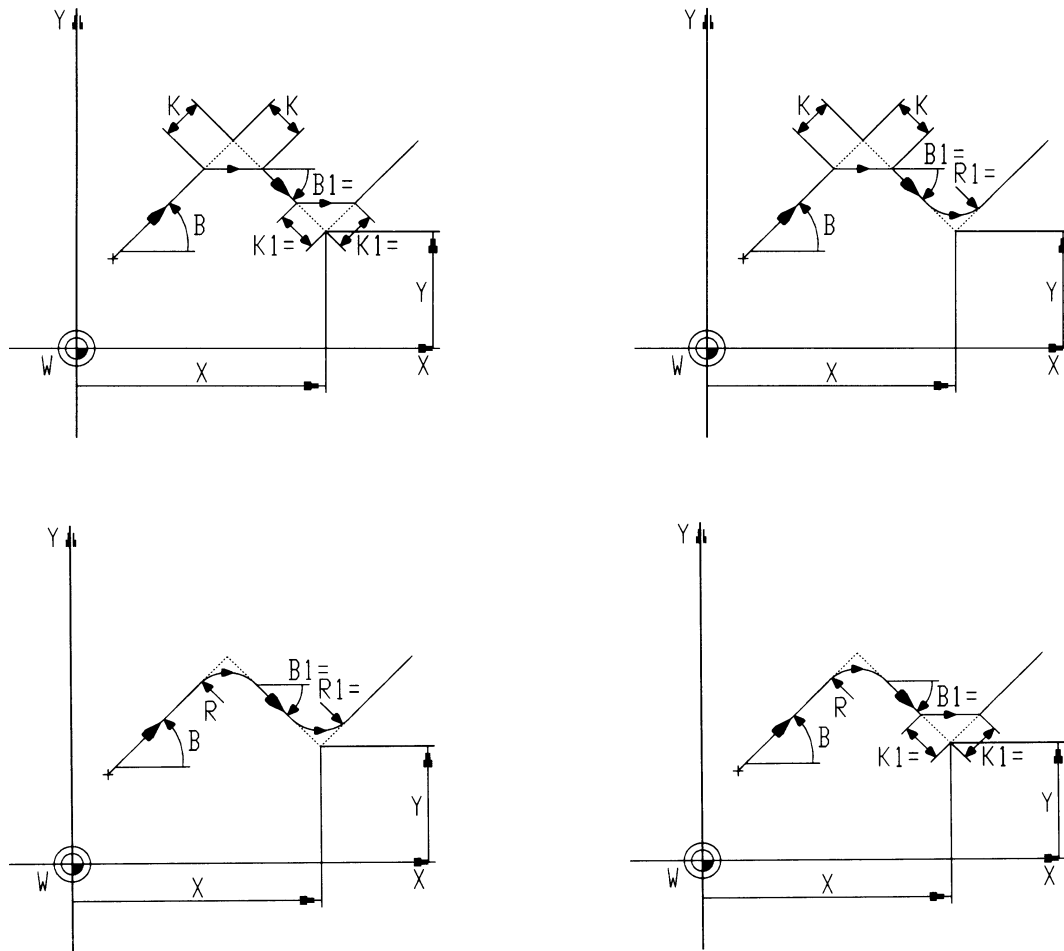
NB8670

Absolute coordinates (G90)



NB8671

Incremental coordinates (G91)



Chamfers or roundings with two-line geometry

Formats

One point geometry (XY-plane)

N... G11 X... Y... {Z...} {K...} {R...} {F...}

N... G11 B... L... {Z...} {K...} {R...} {F...}

Two point geometry (XY-plane)

N... G11 X... Y... X1=... Y1=... {K...} {R...} {K1=...} {R1=...} {F...}

N... G11 B... L... X1=... Y1=... {K...} {R...} {K1=...} {R1=...} {F...}

N... G11 X... Y... B1=... L1=... {K...} {R...} {K1=...} {R1=...} {F...}

N... G11 B... L... B1=... L1=... {K...} {R...} {K1=...} {R1=...} {F...}

Two line geometry (XY-plane)

N... G11 B... X... Y.. B1=... {K...} {R...} {K1=...} {R1=...} {F...}

Parameters

G Linear chamfer rounding cycle
X Endpoint coordinate
Y Endpoint coordinate
Z Endpoint coordinate
B First angle
K First chamfer length
L First length
R First rounding radius
X1= Arbitrary endpoint coordinate
Y1= Arbitrary endpoint coordinate
B1= Second angle
K1= Second chamfer length
L1= Second length
P1= Point definition number

R1= Second rounding radius

One point geometry

X, Y, Z	Linear axis coordinates (absolute/incremental)
P	Point definition number.
P1=	Point definition number.
B	G90 active: Angle the line through the datum point W and the end point makes with the X-axis (G17 and G18) or -Z-axis (G19) G91 active: Angle the line makes with the X-axis (G17 and G18) or -Z-axis (G19)
L	G90 active: length measured from the datum point W to the end point G91 active: length measured from the last tool position to the end point

Two point geometry

X, Y, Z	Linear axis coordinates of the first point (absolute/incremental). No tool axis allowed.
P1=	Point definition number of the first point.
B	G90 active: Angle the line through the datum point W and the first end point makes with the X-axis (G17 and G18) or -Z-axis (G19) G91 active: Angle the first line makes with the X-axis (G17 and G18) or -Z-axis (G19)
L	G90 active: length measured from the datum point W to the first end point G91 active: length measured from the last tool position to the first end point
X1=, Y1=, Z1=	Linear axis coordinates of the second point (absolute/incremental). No tool axis allowed.
P2=	Second point definition number.
B1=	G90 active: Angle the line through the datum point W and the second end point makes with the X-axis (G17 and G18) or -Z-axis (G19) G91 active: Angle the second line makes with the X-axis (G17 and G18) or -Z-axis (G19)
L1=	G90 active: length measured from the datum point W to the second end point G91 active: length measured from the first end point to the second end point

Two line geometry

X, Y, Z	Linear axis end point coordinates of the second line (absolute/incremental). No tool axis allowed.
P	Point definition number of the end point of second line.
P1=	Point definition number of the end point of second line.
B	Angle the first line makes with the X-axis (G17 and G18) or -Z-axis (G19)
B1=	Angle the second line makes with the X-axis (G17 and G18) or -Z-axis (G19)

Words for chamfer or rounding in the three cases

K	First chamfer length
R	First rounding radius
K1=	Second chamfer length
R1=	Second rounding radius

Modal words

F, S, T, H, M, E...=
T1=, T2=

Associated functions

G1

Type of function

Non-modal

Notes and usage**FEEDRATE**

All movements in a G11-block use the last programmed feedrate if a
New feedrate is not stated in the G11-block.

NEXT MOVEMENT AFTER A G11 BLOCK

If a second chamfer (K1=) or a second rounding (R1=) has been programmed, the
block following the G11-block must contain either a G1 or G11-function.

If a G1-block has been programmed following a G11-block, both end point
coordinates (e.g. X. and Y.) must be stated.

TOOL AXIS PROGRAMMING WITH A G11

With the one point geometry it is allowed to program the tool axis with the end point
coordinates. The three-axis movement is executed before making the chamfer or
rounding programmed in the same G11-block.

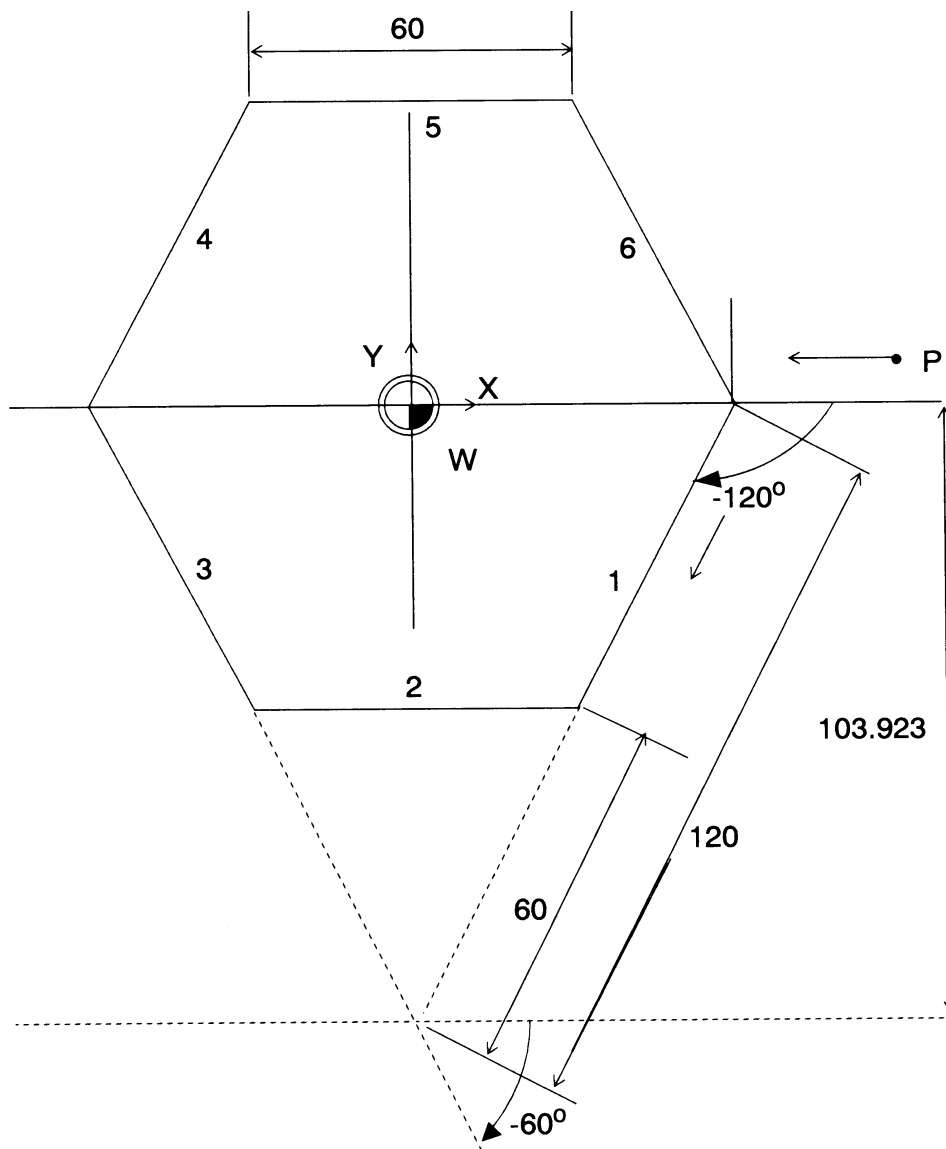
With the two points or two-line geometry tool axis programming is not allowed

RESTRICTION

1. The G11 function is not allowed once the geometry is activated (G64 active).
2. The G11 function is not allowed for defining a pocket or island contour. (G200..G208)
3. G11 is not allowed with a programmed tool axis. In case programs are started with above mentioned programming combination, the programmer may encounter operational errors P01 and/or P34 at execution of the program.

Examples

EXAMPLE 1. One point geometry



NB8579

The regular hexagon has to be milled on the outside of the workpiece surface. The one point geometry with angle is used. The sides 2 and 4 are programmed as chamfers.

N9010
N1 G17 T1 M6
N2 G0 X100 Y10 Z-10 S1000 M3
N3 G1 F300
N4 G43 X60
N5 G41 Y0
N6 G11 B-90 L103.923 K60
N7 G11 B150 L103.923 K60
N8 G11 B60 L60
N9 G11 B0 L60
N10 G40
N11 G1 X100 Y10
N12 G0 Z100 M30

Explanation

N9010: Program number.

N1: Activate the main plane. Load the tool

N2: Start the spindle, move tool to point P and to depth.

N3: Set feedrate to 300 mm/min.

N4: Move the tool to the corner of the hexagon.

N5: Set radius compensation LEFT.

N6: Mill along sides 1 and 2. Programmed is the intersection point of sides 1 and 3 the chamfer (K-word) around this point.

N7: Mill along sides 3 and 4. Programmed is:

- The intersection point of sides 3 and 5
- The chamfer (K-word) around this point.

N8: Mill along side 5.

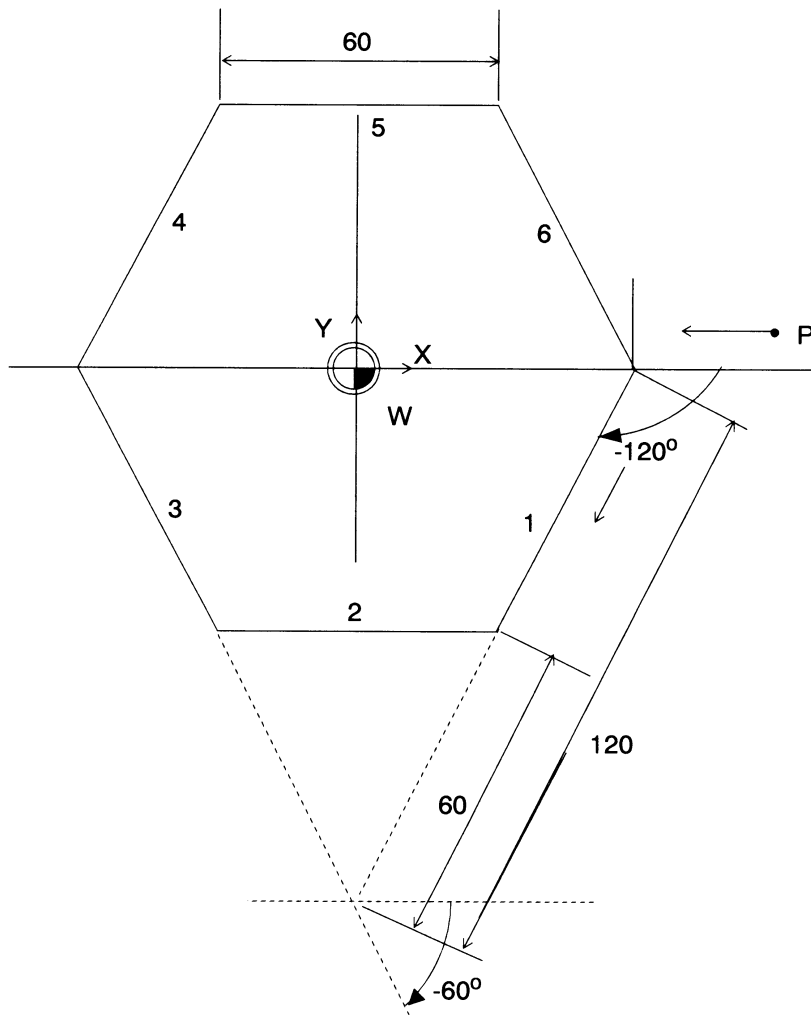
N9: Mill along side 6.

N10: Cancel the radius compensation.

N11: Move tool away from part.

N12: Retract the tool and end of program.

EXAMPLE 2. Two point geometry



NB8580

The regular hexagon has to be milled on the outside of the workpiece surface. The two-point geometry with angles and increments is used. The sides 2 and 5 are programmed as chamfers.

```

N9011
N1 G17 T1 M6
N2 G0 X100 Y10 Z-10 S1000 M3
N3 G1 F300
N4 G43 X60
N5 G41 Y0
N6 G91
N7 G11 B-120 L120 K60 B1=-120 L1=120
N8 G11 B60 L120 K60 B1=-60 L1=120
N9 G40
N10 G90
N11 G1 X100 Y10
N12 Z10 M30

```

Explanation:

N9011: Program number.

N1: Activate the main plane. Load tool 1

N2: Start the spindle, move tool to point P and then to depth.

N3: Enter the linear movement and set the feedrate.

N4: Move the tool to the corner of the hexagon.

N5: Set radius compensation LEFT.

N6: Activate the incremental mode. The length values in the next blocks are measured from the previous tool position.

N7: Mill along sides 1, 2 and 3. Programmed is

- The intersection point of sides 1 and 3 (B and L),
- The chamfer (K-word) around this point
- The end point of side 3 (B1= and L1=).

N8: Mill along sides 4, 5 and 6. Programmed is

- The intersection point of sides 4 and 6 (B and L),
- The chamfer (K-word) around this point
- The end point of side 6 (B1= and L1=)

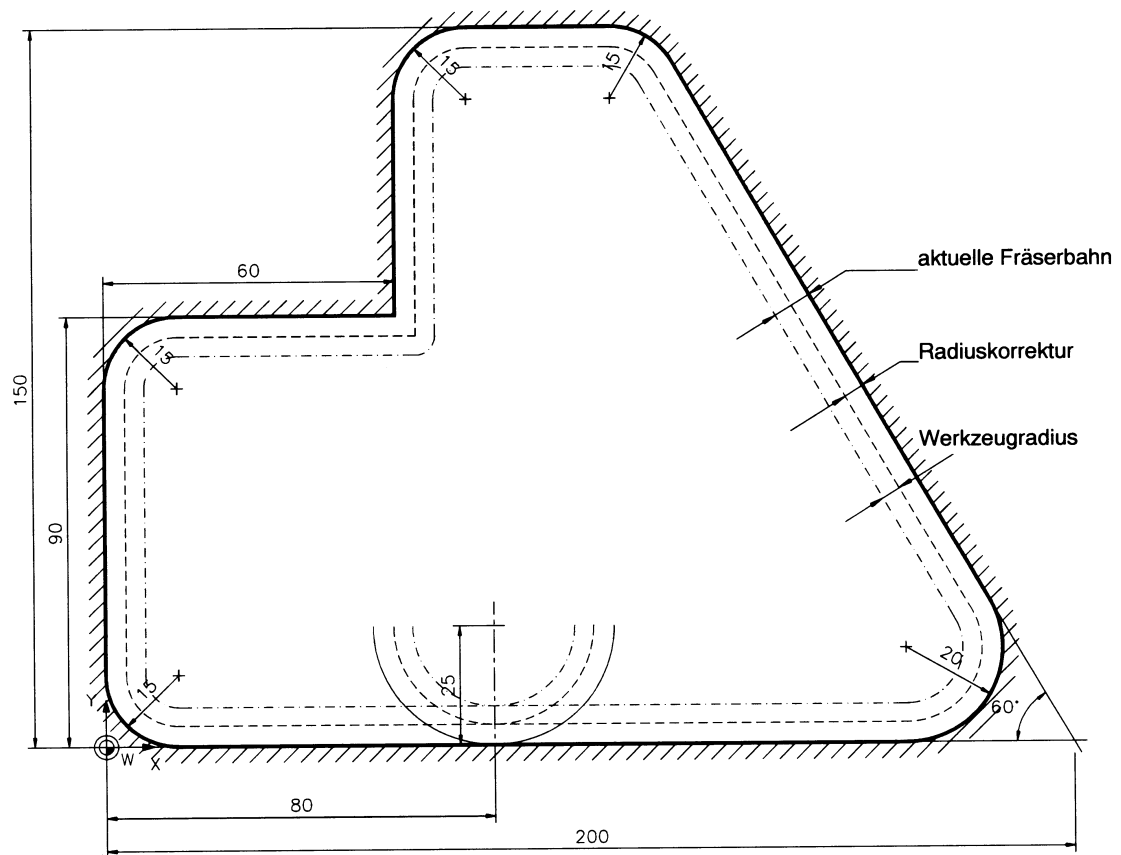
N9: Cancel radius compensation.

N10: Set the absolute mode.

N11: Move tool away from workpiece.

N12: End of program.

EXAMPLE 3. Two line geometry



NB8676

The inside pocket can be programmed using G11-function with two line geometry elements.

```

N9012
N1 G17
N2 X80 Y25 Z0 T1 M6
N3 G1 Z-10 F500 S1000 M3
N4 G43 X105
N5 G42
N6 G2 X80 Y0 R25 F300
N7 G11 X0 Y90 B180 B1=90 R15 R1=15
N8 G11 X60 Y150 B0 B1=90 R1=15
N9 G11 X200 Y0 B0 B1=120 R15 R1=20
N10 G1 X80 Y0
N11 G2 X55 Y25 R25
N12 G40
N13 G0 Z200 M30

```

Explanation:

- N9012: Program number.
- N1: Activate XY-plane (G17)
- N2: Load tool T1 (Mill diameter 10 mm). Move the tool to point B and above the workpiece.
- N3: Start the spindle and feed to depth.
- N4: Move the tool to the starting point of the entering-circle.
- N5: Set radius compensation RIGHT.
- N6: Move to the contour via the entering-circle.
- N7: Mill - along the X-axis, (B0)
 - Along the radius, (R15)
 - Along the Y-axis, (B1 =90)
 - Along the second radius (R1 =15).
- N8: Mill - parallel to the X-axis, (B0)
 - Parallel to the Y-axis, (B1 =90)
 - Along the second radius (R1 =15)
- N9: Mill - parallel to the X-axis, (B0)
 - Follow the first radius, (R15)
 - Mill along the slope of 60 degrees (B1=120)
 - Follow the second radius (R20).
- N10: Mill along the X-axis to the starting point of the circle for leaving the contour.
- N11: Exit the contour with a circular movement.
- N12: Cancel the radius compensation.
- N13: Retract the tool. End of program.

5.10 G14 Repeat function

Purpose

To repeat the execution of a specified number of blocks within a partprogram or subprogram.

Format

N... G14 N1=... {N2=...} {J...} {K...}

Parameters

G	Repeat funktion
J	Number of repeats
K	Repeat decrement
N1=	Repeater begin block
N2=	Repeater end block

Associated functions

G22, G29

Type of function

Non-modal

Notes and usage

BLOCK NUMBERS OF REPEAT SEQUENCE (N1=, N2=)

These block numbers must be in the same partprogram or subprogram.

If N2= is not programmed, only the block indicated by N1 = is repeated the specified number of times.

ORDER OF BLOCKS TO BE REPEATED

The order of executing the blocks in the repeat sequence must be the same as the order originally programmed. So in the program block N1=.. must be before block N2=..

NUMBER OF REPEATS (J)

The number of repeats is programmed with the J-word. The J-word is not necessarily an integer value. The integer part, thus the part before the decimal point, is used as the number of repeats.

When no number of repeats is programmed (no J-word is present), the sequence is repeated only once.

REPEAT DECREMENT (K)

The K-word allows the value of the J-word to be recalculated and used as the condition for repeating.

If the K-word is not programmed, the value of the J-word is reduced by 1 after every repeat.

If $K > 0$, the value is used to reduce the value of the J-word. If e.g. K5 were programmed, 5 would be subtracted from the value of the J-word after every repeat. As long as the J-word is greater than 0, a repeat is executed.

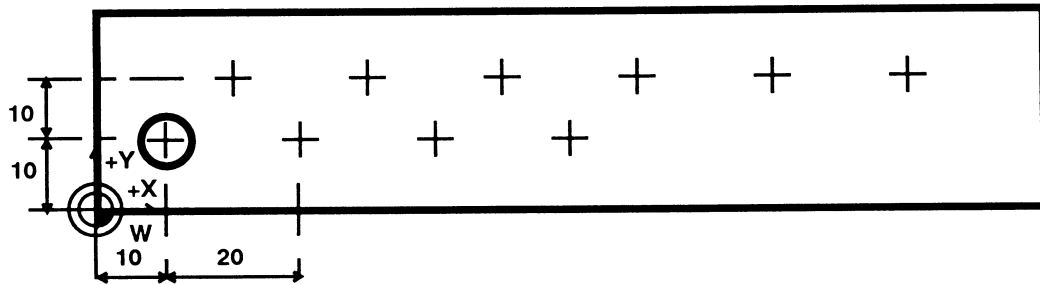
If $K \leq 0$ is programmed, an error message is displayed.

NESTING OF REPEATS

A repeating block sequence may be included in another repeating block sequence; this can be done four times.

CONTINUATION AFTER THE REPEAT

Once the repeats are executed, the program continues with the block after the G14.

Example**EXAMPLE 1.** Programming a repeat function

```

N1234
N1 G195 X-10 Y-10 Z10 I160 J50 K-30
N2 G99 X0 Y0 Z0 I140 J30 K-10
N3 G17
N4 T1 M6
N5 G81 Y5 Z-11.5 F0.2 S2000 M3
N6 G79 X10 Y10 Z0
N7 G79 L1=20 B1=0
N8 G14 N1=7 J2
N9 G92 X10 Y10
N10 G14 N1=6 N2=8
N11 G14 N1=7 N2=7 J2
N12 G93 X0 Y0
N13 G0 Z200
N14 M30

```

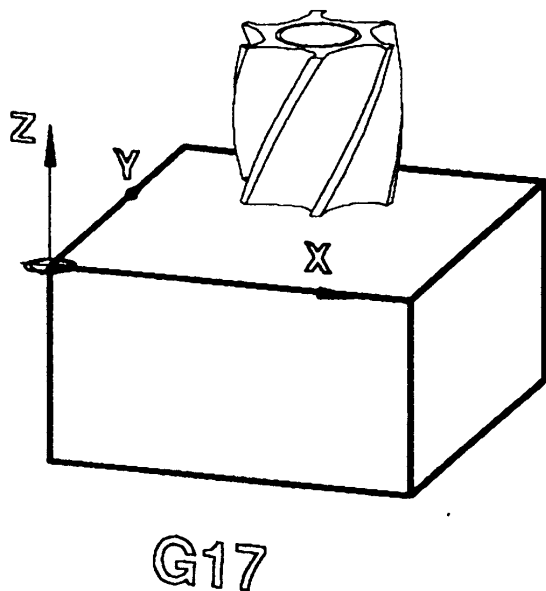
Explanation:

N1: Set up graphic window
 N2: Set up graphic (material)
 N3: Define the main plane.
 N4: Load tool 1 with a drill diameter of 10 mm.
 N5: Define fixed drilling cycle and start the spindle
 N6: Drill holes
 N7: Drill holes with Polar coordinates.
 N8: Program block N7..N8 will be repeated twice.
 N9: Absolute zero point shift
 N10: Program block N7..N8 will be repeated once.
 N11: Program block N7 will be repeated twice.
 N12: Absolute zero point shift.
 N13: Retract the tool to Z200.
 N14: End of program.

5.11 G17 Mainplane XY, tool Z

The main spindle of the machine tool determines the position of the tool axis. With G17 is defined that the tool axis is the Z-axis and the main plane for milling operations the XY-plane.

For turning mode refer to chapter 12.4 "Mainplane for turning mode" on page 556.



Format

N... G17

Parameters

None

Modal words
H, T1=

Associated Functions

G18/G19

Type of function

Modal

Notes and usage

DEFAULT PLANE

When switching on the machine or after a CLEAR CONTROL the machine activates automatically due to MC11 (0=G17, 1=G18, 2=G19) a PLANE.

The last selected PLANE is active when the machine is switch on normally.

OPERATIONS IN THE PLANE

Calculations for radius compensation, the geometry (G64), polar coordinate, milling cycles; the pocket cycle, etc. are performed in the current plane. Thus when G17 is active the XY-plane.

OPERATIONS IN THE TOOL AXIS

Tool length compensation and the fixed cycles for hole operations use the current tool axis. Thus when G17 is active the Z-axis.

ANGULAR HEAD

When an angular head is fitted, the axis configuration of the machine tool remains unchanged. So the tool can be in either the Y- or X-axis.

With the function G18 or G19 is programmed in which axis the tool is standing and which plane is the plane of operation.

G18: XZ-plane, tool in Y-axis

G19: YZ-plane, tool in X-axis

Refer to G19 using an angular head for programming an angular head.

CHANGING THE PLANE OF OPERATION

When a new plane is selected, thus either G18 or G19 activated, the length compensation in the Z-axis is cancelled and activated in the tool axis related to the selected plane.

CANCELLATION

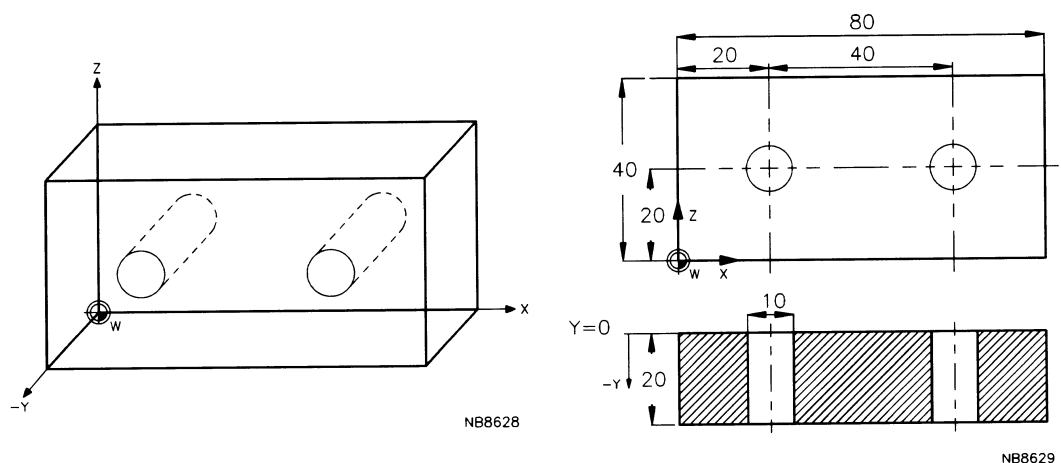
The G17 function is cancelled by activating another machining plane, using either the G18 or G19 functions. The G17 function is not cancelled by CLEAR CONTROL or by softkey CANCEL PROGRAM.

TOOL OFFSETS

Tool dimensions stored in the Tool Memory are independent of the selected plane.

Example

EXAMPLE 1: G17



```

N9001
N1 G17
N2 T1 M6
N3 G0 X20 Y20 Z1 F400 S1600 M3
N4 G1 Z-23.5
N5 G0 X60 Z1
N6 G1 Z-23.5
N7 G0 Z200 M30

```

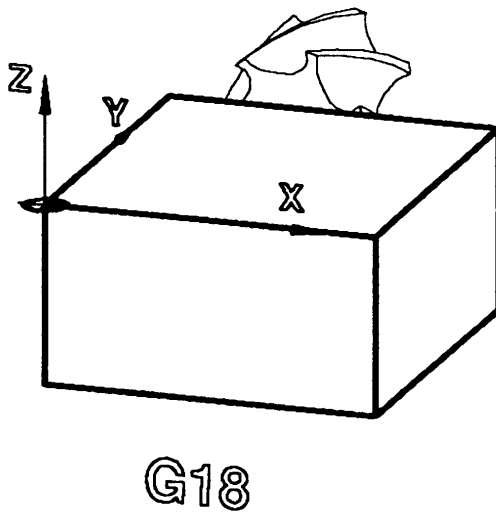
Explanation:

- N1: Activate G17 main plane.
- N2: Load tool T1 and its offsets. Drill diameter is 10 mm.
- N3: Move tool rapidly (G0) to programmed position. Set feedrate to 400 mm/min. Make spindle Rotate clockwise (M3) at 1600 rev/min.
- N4: Feed tool to programmed depth.
- N5: Retract tool to Z1 and then move the tool rapidly to X60. The CNC's positioning logic ensures That the tool does not collide with the workpiece, because the tool is first moved along the Z-axis before moving along the X-axis.
- N6: Feed tool to programmed depth.
- N7: Retract tool to Z200 and end of program.

5.12 G18 Mainplane XZ, tool Y

Purpose

The main spindle of the machine tool determines the position of the tool axis. With G18 is defined that the tool axis is the Y-axis and the main plane for milling operations the XZ-plane.



Format

N... G18

Parameters

None

Modal words
H, T1=

Type of function

Modal

Associated Functions

G17/G19

Notes and usage

DEFAULT PLANE

When switching on the machine or after a CLEAR CONTROL the machine activates automatically due to MC11 (0=G17, 1=G18, 2=G19) a PLANE.

The last selected PLANE is active when the machine is switch on normally.

OPERATIONS IN THE PLANE

Calculations for radius compensation, the geometry (G64), polar coordinate, milling cycles; the pocket cycle, etc. are performed in the current plane. Thus when G18 is active the XZ-plane.

OPERATIONS IN THE TOOL AXIS

Tool length compensation and the fixed cycles for hole operations use the current tool axis. Thus when G18 is active the Y-axis.

ANGULAR HEAD

When an angular head is fitted, the axis configuration of the machine tool remains unchanged. So the tool can be in either the Z- or X-axis.

With the function G17 or G19 is programmed in which axis the tool is standing and which plane is the plane of operation.

G17: XY-plane, tool in Z-axis

G19: YZ-plane, tool in X-axis

Refer to G19 using an angular head for programming an angular head.

CHANGING THE PLANE OF OPERATION

When a new plane is selected, thus either G17 or G19 activated, the length compensation in the Y-axis is cancelled and activated in the tool axis related to the selected plane.

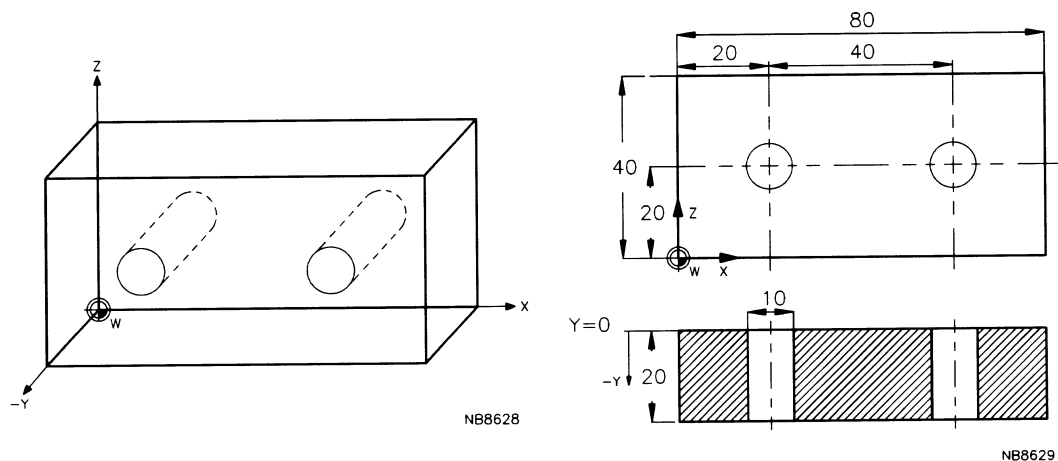
CANCELLATION

The G18 function is cancelled by activating another machining plane, using either the G17 or G19 functions. The G18 function is not cancelled by CLEAR CONTROL or by softkey CANCEL PROGRAM.

TOOL OFFSETS

Tool dimensions stored in the Tool Memory are independent of the selected plane.

Example



```

N9002
N1 G18
N2 T2 M6
N3 G0 X20 Y1 Z20 F400 S1600 M3
N4 G1 Y-23.5
N5 G0 X60 Y1
N6 G1 Y-23.5
N7 G0 Y200 M30

```

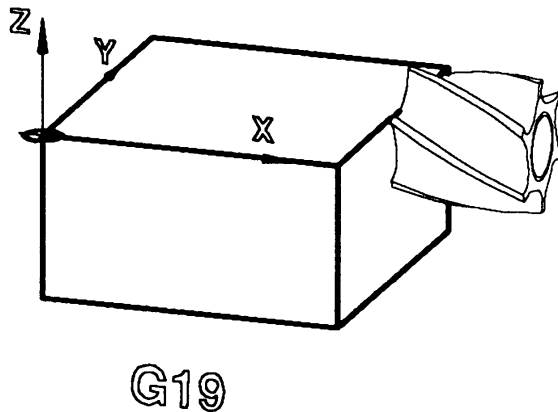
Explanation:

- N1: Make XZ-plane (G18) active.
- N2: Load tool T2 and its offsets. Drill diameter is 10 mm.
- N3: Move tool rapidly (G0) to the programmed position. Set the feedrate to 400 mm/min and Make spindle rotate clockwise (M3) at 1000 rev/min.
- N4: Feed tool to depth.
- N5: Retract tool to Y1 and then move tool rapidly to X60. The CNC's positioning logic ensures that The tool does not collide with the workpiece, because the tool is first moved along the Y-axis before moving along the X-axis.
- N6: Feed tool to depth.
- N7: Retract tool to Y200 and end of program.

5.13 G19 Mainplane YZ, tool X

Purpose

The main spindle of the machine tool determines the position of the tool axis. With G19 is defined that the tool axis is the X-axis and the main plane for milling operations the YZ-plane.



Format

N... G19

Parameters

None

Modal words

H, T1=

Associated Functions

G17/G18

Type of function

Modal

Notes and usage

DEFAULT PLANE

When switching on the machine or after a CLEAR CONTROL the machine activates automatically due to MC11 (0=G17, 1=G18, 2=G19) a PLANE.

The last selected PLANE is active when the machine is switch on normally.

OPERATIONS IN THE PLANE

Calculations for radius compensation, the geometry (G64), polar coordinate, milling cycles; the pocket cycle, etc. are performed in the current plane. Thus when G19 is active the YZ-plane.

OPERATIONS IN THE TOOL AXIS

Tool length compensation and the fixed cycles for hole operations use the current tool axis. Thus when G19 is active the X-axis.

ANGULAR HEAD

When an angular head is fitted, the axis configuration of the machine tool remains unchanged. So the tool can be in either the Z- or Y-axis.

With the function G17 or G18 is programmed in which axis the tool is standing and which plane is the plane of operation.

G17: XY-plane, tool in Z-axis

G18: XZ-plane, tool in Y-axis

Refer to using an angular head for programming an angular head.

CHANGING THE PLANE OF OPERATION

When a new plane is selected, thus either G17 or G18 activated, the length compensation in the X-axis is cancelled and activated in the tool axis related to the selected plane.

CANCELLATION

The G19 function is cancelled by activating another machining plane, using either the G17 or G18 functions. The G19 function is not cancelled by CLEAR CONTROL or by softkey CANCEL PROGRAM.

TOOL OFFSETS

Tool dimensions stored in the Tool Memory are independent of the selected plane.

USING AN ANGULAR HEAD

When an angular head is used, its dimensions must be programmed. Either a zero point shift (G92 or G93) or a stored zero offset (G54 - G59) can be used for this purpose. The use of stored zero offsets is recommended because the partprogram remains independent of the dimensions of the angular head.

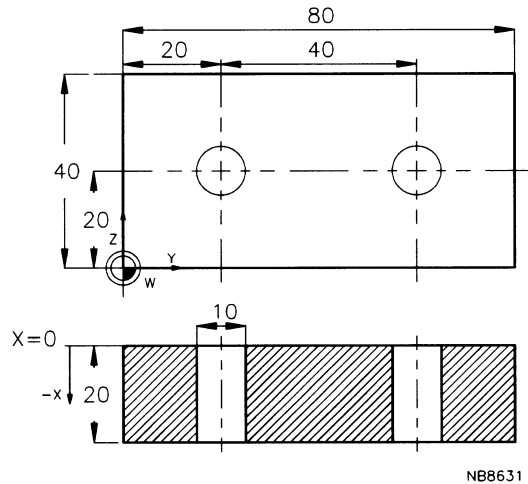
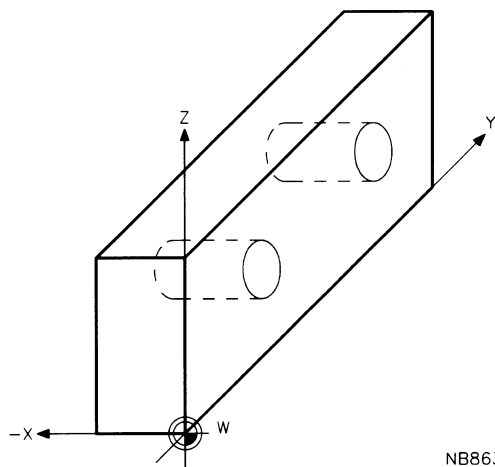
TOOL IN + OR - DIRECTION OF X-AXIS

Especially with an angular head in the X-axis the tool can be in the positive (+) or negative (-) direction of the axis. The functions G66 and G67 are available to indicate in which direction the tool is standing and allows the partprogrammer to look always in the same way at the plane of operation.

Refer to G66/G67 for using these functions.

Example

EXAMPLE 1: A vertical milling machine is assumed.



```

N9003
N1 G19
N2 T3 M6
N3 G0 X1 Y20 Z20 F400 S1600 M3
N4 G1 X-23.5
N5 G0 X1 Y60
N6 G1 X-23.5
N7 G0 X200 M30

```

Explanation:

- N1: Make YZ-plane (G19) active.
- N2: Select tool T3 and its offsets. Drill diameter is 10 mm.
- N3: Move tool rapidly (G0) to programmed position. Set the feedrate to 400 mm/min and make Spindle rotate clockwise (M3) at 1000 rev/min.
- N4: Feed tool to depth.
- N5: Retract tool to X1 and then move tool rapidly to Y60. The CNC's positioning logic ensures that The tool does not collide with the workpiece, because the tool is first moved along the X-axis before moving along the Y-axis.
- N6: Feed tool to depth.
- N7: Retract tool to X200 and end of program.

5.14 G22 Macro call

Purpose

To execute a subprogram with standard operations.

Format

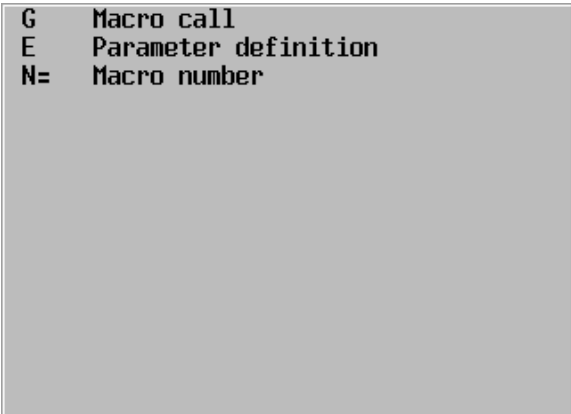
To call a subprogram.

N... G22 N=... {E...=}

To activate a subprogram on the condition that E...>0

N... G22 E... N=... {E...=}

Parameters



```
G   Macro call
E   Parameter definition
N=  Macro number
```

Associated functions

G14, G29

Type of function

Non-modal.

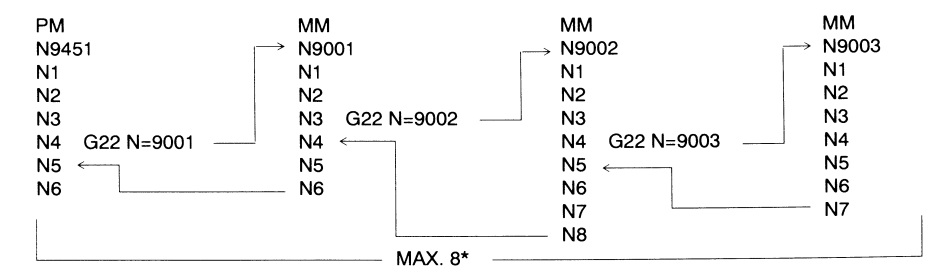
Notes and usage

ACTIVATION

A subprogram is completely executed when it is called from a main program or from another subprogram.

NESTING OF SUBPROGRAMS

When a subprogram calls another subprogram, the subprogram, which is activated, is referred to as a 'nested' subprogram. At the end of a nested subprogram the calling subprogram continues. A maximum of eight 'nested' programs can be used.



E-PARAMETERS

A subprogram may contain E-parameters, which are variables whose values are stored in a separate CNC memory. Subprograms can therefore be written which have a general application. When the dimensions of a component are known, only the E-parameter values need to be altered, not the program.

E-parameters can get their value in the main program or subprogram, via the operator's panel, or by reading-in the parameter memory.

Arithmetical calculations with parametric values are allowed in programs and subprograms. The same parameter can be used by different (sub) programs.

Refer to the special appendix about E-parameters at the end of this manual for more details of programming with E-parameters.

NUMBER OF PARAMETER DEFINITIONS

In a block with a macro call up to 10 parameters can get their value. If more parameters are used, extra lines before the macro call are necessary.

EVALUATION OF DEFINED PARAMETERS

Any value or arithmetical expression can be assigned to a parameter in a G22-block.

Parameters programmed in the G22 block are evaluated and calculated before the execution of the macro.

CONTINUATION AFTER THE MACRO CALL

Once the macro is executed, the program continues with the block after the G22 in which the macro was called.

CONDITIONAL MACRO CALL (E)

The value of the E-word is used for dictating if a conditional macro call must be performed.

If the value of E...>0, the macro call is performed.

After the call the program continues with the block after the G22. Parameter E... is not influenced by the macro call.

If the value of E...<=0, the macro is not called. The program continues with the block after the G22.

Examples

EXAMPLE 1. A macro call.

N100 G22 N=9100 E1=24 E2=3

Explanation:

N150: Execute subprogram N9100 with parameters E1=24 and E2=3.

EXAMPLE 2. Conditional macro call.

N150 G22 E60 N=9100

Explanation:

N150: Execute subprogram N9100 when the value of E60 > 0.

EXAMPLE 3. Macro without E-parameters.

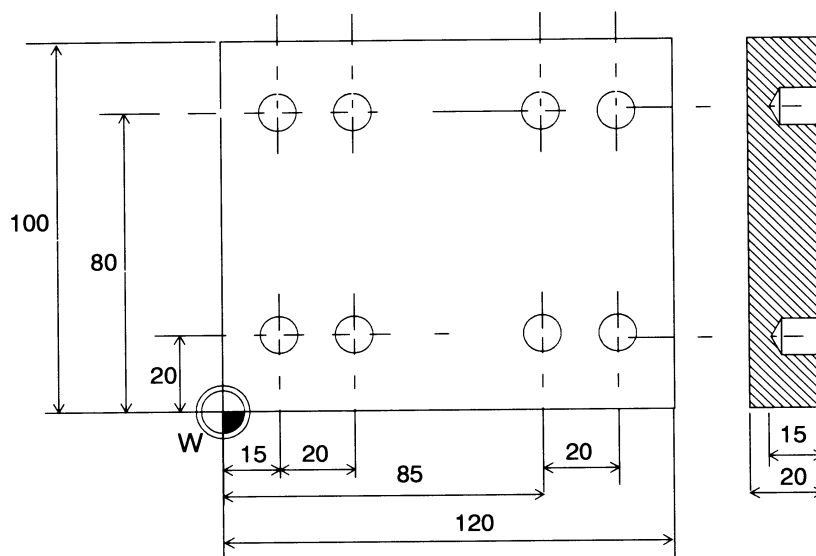
Subprogram for drilling two holes:

```
N9001
N1 G91
N2 G1 Z-16 M8
N3 G0 Z16 M9
N4 X20
N5 G1 Z-16 M8
N6 G0 Z16 M9
N7 G90
```

Explanation of the macro:

N1: Activate incremental programming.
N2: Switch coolant ON. Move tool with a feedrate in negative direction.
N3: Retract tool. Switch coolant OFF.
N4: Move tool 20 mm along X-axis to second start position.
N5: Switch coolant ON. Feed tool 15 mm into workpiece.
N6: Retract tool. Switch coolant OFF.
N7: Re-activate absolute programming.

EXAMPLE 4: Main program for drilling four pairs of holes:



NB8537

Main program

```

:
N45 T1 M6
N50 F400 S1600 M3
N55 G0 X15 Y20 Z1
N60 G22 N=9001
N65 G0 X85
N70 G22 N=9001
N75 G0 X85 Y80
N80 G22 N=9001
N85 G0 X15
N90 G22 N=9001

```

Macro program

```

N9001
N1 G91
N2 G1 Z-15
N3 G0 Z16
N4 G90

```

Explanation of the main program:

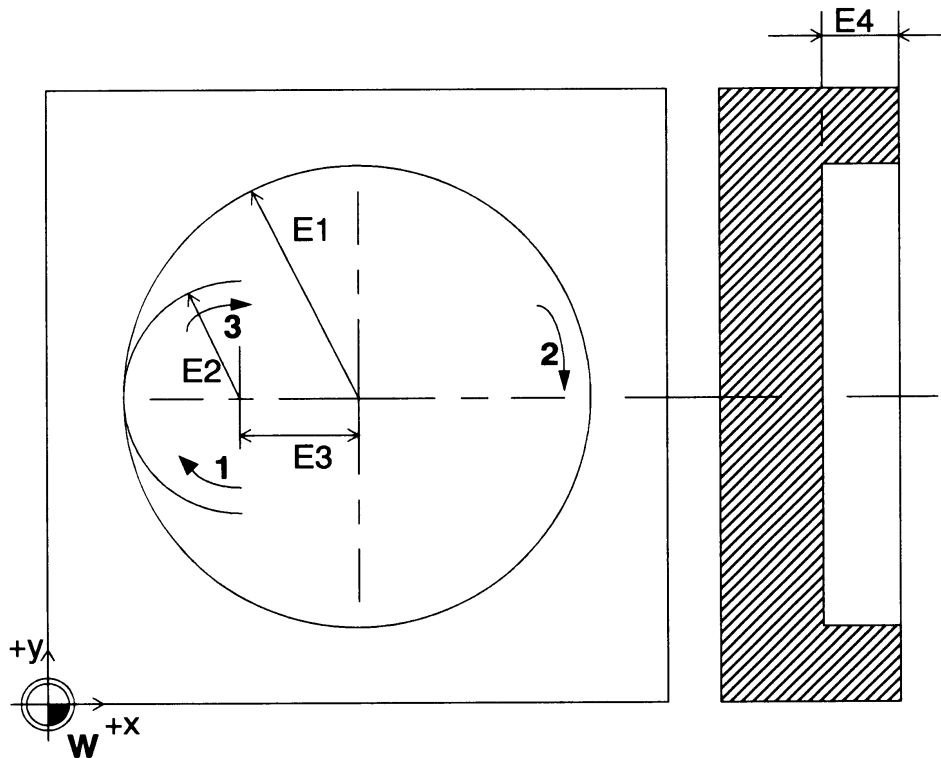
N45: Load tool T1 and use its offsets. Drill diameter is 10 mm.
 N50: Make spindle rotate clockwise at 1600 rev/min. Feedrate at 400 mm/min
 N55: Move tool to first drilling position and 1 mm off top surface.
 N60: Activate subprogram
 N65: Move tool to second drilling position.
 N70: Activate subprogram
 N75: Move tool to third drilling position.
 N80: Activate subprogram
 N85: Move tool to fourth drilling position.
 N90: Activate subprogram

Programming instruction

G0 is not really necessary in N65, N75 and N85 blocks, because G0 is programmed in the last block of the subprogram. G0 has been programmed in the latter block, because then you need not know how exactly the subprogram ends in order to understand the main program.

EXAMPLE 5. Subprogram, which uses E-parameters.

This example describes a general subprogram for finishing a milled full circle. To achieve a smooth transition the circle is entered and exited with a small circular arc. Tool radius compensation is used so that the workpiece contour can be programmed directly.



NB7973

The following parameters must be set in the program:

- E1: Radius R1 of the full circle
- E2: Radius R2 of the approach and retraction circle
- E4: The depth, including a clearance distance.

The following parameter is calculated:

- E3: The distance (R1-R2).

Programming instruction

Make the subprogram as universal as possible, i.e. without tool change, feed and speed. You will then also be able to use the subprogram for other tools. Set the tool change, feed and speed in the main program.

The subprogram is written with incremental dimensions.

```

N9002
N1 G91
N2 E3=E1-E2
N3 G1 Z=-E4
N4 X=-E3
N5 G43 Y=-E2
N6 G42
N7 G2 X=-E2 Y=E2 R=E2
N8 I=E1 J=0
N9 X=E2 Y=E2 R=E2
N10 G40
N11 G0 X=E3 Y=-E2 Z=E4
N12 G90

```

Explanation:

N1: Incremental programming is set.
 N2: Parameter E3 is calculated.
 N3: Feed tool to depth. In E4 a clearance from the workpiece must be considered.
 N4: Move tool to the centre of the approach circle.
 N5: Feed tool to the starting point of the approach circle.
 N6: Set radius compensation RIGHT.
 N7: Move tool in a clockwise direction to produce an arc. Both the end point and the radius are programmed with parameters.
 N8: Mill the complete circle. The circle centre is programmed.
 N9: Mill the area of the retraction circle. Both the end point and the radius are programmed with parameters.
 N10: Cancel the radius compensation.
 N11: Move tool back to the centre of the circle and retract at rapid traverse rate.
 N12: Re-activate absolute programming (G90).

When this subprogram is used the calling program must contain:

- A tool movement to the circle centre (N200)
- A definition of the parameters (N201)
- A call of the subprogram (N9002)

The calling program is as follows:

```

N9002
N1 G17
N2 G54
N3 T1 M6
N4 F900 S2000 M3
:
N200 G0 X75 Y80 Z0
N201 G22 N=9002 E1=30 E2=15 E4=15
:
N500 M30

```

After calling the subprogram, a circle with a diameter of 60 mm is milled in the required (X75, Y80)

5.15 G23 Main program call

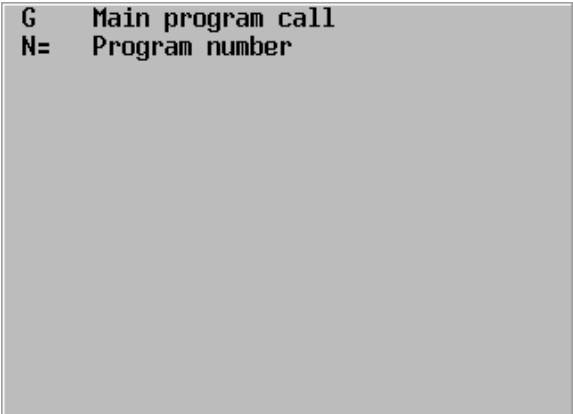
Purpose

To call a partprogram from a main program.

Format

N... G23 N=... {E...=...}

Parameters



```
G   Main program call
N=  Program number
```

Associated functions

G14, G22, G29

Type of function

Non-modal.

Notes and usage

ACTIVATION

A called partprogram is completely executed when it is called from a main program.

PROGRAM SIZE

Programs smaller than 100 Kbytes will be stored in the work-memory and executed as a normal G23 call.

Programs bigger than 100 Kbytes cannot be stored in the work-memory. They will be separated automatically and invisible, in a lot smaller partprograms. These partprograms will be executed automatically (CAD MODE).

CONDITIONAL JUMPS

In the main program conditional jumps can be used to decide which program should be executed.

RESTRICTIONS

A called partprogram cannot contain a G23-function; partprograms cannot be "nested" within one another.

A subprogram (macro) must not contain the G23-function.

Programs bigger than 100 Kbytes may not have jump instructions.

CONTINUATION AFTER THE PROGRAM CALL

Once the called program is executed, the main program continues with the block after the G23 in which the program was called.

TERMINATION OF A CALLED PROGRAM

When the execution of a called partprogram will be stopped with Intervention or softkey <Cancel program>, a jump to the mainprogram start will be done.

Example

```

N9990
:
N10 G23 N=988      Activate macro program
                   N988
                   N1
                   N2
                   :
                   :
                   N200 M30
N11               Jumping back to main program.
N20 G23 N=989
:
N30 M30

```

Explanation

N10: Program N988 is called

N20: Program N989 is called

5.16 G25/G26 Enable/Disable feed- and/or speed-override

Purpose

To enable or disable the feed- and/or speed-override, in order to control programmed feed and speed movements. If the feed- or speed-override is disabled, the feed- or speed-override is fixed to 100%.

Format

To enable feed- and speed-override:
N... G25

To disable feed override (F=100%):
N... G26 I2=1 or without I2

To disable speed override (S=100%):
N... G26 I2=2

To disable feed- and speed-override (F and S=100%):
N... G26 I2=3

Parameters

G Disable feed/speed override
I2= 1=F100%; 2=S100%; 3=F+S100%

Type of function

Modal

Notes and usage

DEFAULT MODE

The CNC system automatically activates G25 at the start of a partprogram.

CANCELLATION

The G26-function is cancelled by G25 or with softkey CLEAR CONTROL or softkey CANCEL PROGRAM or M30.

Example

N66 G26 I2=1 : Feed override switched off, thus F fixed to 100%.
N67 G26 I2=2 : Speed override switched off, thus S fixed to 100%.
N68 G26 I2=3 : Feed- and speed-override switched off, thus F and S fixed to 100%.
N70 G25 : Feed override switched on.

5.17 G27/G28 Positioning functions

Purpose

1. For indicating with feed movements (G1, G2/G3, G6) when the next movement starts and if a stop between the movements should occur.
2. For indicating with a rapid movement (G0) if a stop between the movements should occur or must be avoided. Parameter I4=.
3. For switching off and on the positioning logic with G0 movements. Parameter I5=.
4. Controlling contour tolerance. Parameter I7=.

Note: The Look Ahead Feed-Function (LAF) function controls the path with the highest possible accuracy (<10 µm). However, high accuracy is at the expense of path speed, because the control has to decelerate at each edge to guarantee the contour tolerance. In certain cases (e.g. when roughing), speed is more important than accuracy and this function is not wanted.

5. Affecting of the acceleration. Parameter I6=

Format

To activate:

N... G28 {I3=...} {I4=...} {I5=...} {I6=...} {I7=...}

To cancel each possibility separately

N... G28 {I3=0} {I4=0} {I5=0} {I6=0} {I7=0}

To cancel all possibilities (default setting):

N... G27

Parameters

G27 none

G28:

```
G   Positioning functions
I3= Feed movement 0=inpos, 1=inpod
I4= Rapid movement 0=inpod, 1=inpos
I5= Position logic: 0=with, 1=without
I6= Acceleration reduction      [%]
I7= Contour tolerance
```

Modal words

H, T1=

Type of function

Modal

Associated functions

G0, G1, G2/G3

Movements**MOVEMENTS WITH INPOSITION**

A movement with inposition means that the next movement starts once all programmed axes have reached their programmed position. A stop occurs between the movements.

MOVEMENTS WITHOUT INPOSITION

A movement without inposition means, that the next movement starts as soon as the interpolator of the CNC has reached the commanded position. No deceleration or axes lag is taken into account. There is no stop between the movements, so the program is executed faster, but with feed movements rounding of corners occurs.

Feed movements**FEED MOVEMENTS (G1, G2/G3)**

Parameter I3 controls the point at which the next programmed movement starts after a feed movement.

G28 I3=0

The feed movement is executed WITHOUT INPOSITION. There is no stop between the movements and therefore corners are rounded; machining quality is good.

I3=0 is the default setting with feed movements.

G28 I3=1

The feed movement is executed WITH INPOSITION. A stop occurs between the movements. Corners are sharp, but machining quality is poor.

Rapid traverse movements**RAPID TRAVERSE MOVEMENTS (G0)**

Parameter I4 controls the point at which the next programmed movement starts after a rapid movement (G0).

G28 I4=0

The rapid movement is executed WITH INPOSITION. A stop occurs between the movements.

I4=0 is the default setting with rapid movements.

G28 I4=1

The rapid movement is executed WITHOUT INPOSITION. There is no stop between the movements.

SWITCHING ON/OFF THE POSITIONING LOGIC WITH G0

Parameter I5 indicates if the positioning logic in a G0-block should be executed or switched off. Refer to the function G0 for a description of the positioning logic.

G28 I5=0

G0 is executed with positioning logic.
I5=0 is the default setting.

G28 I5=1

Positioning logic is not active with a G0 movement

Note: The positing logic with a G79 block cannot be switched off.

Programmable acceleration reduction

The acceleration is normally determinate by the acceleration reduction factor per axes (MC3*04). This acceleration will be multiplied with the acceleration reduction. Value between 5 and 100 can be given (5 is a very small acceleration, 100 is normal).

This reduction is active for G0, G1, G2 and G3, which are executed with LAF.

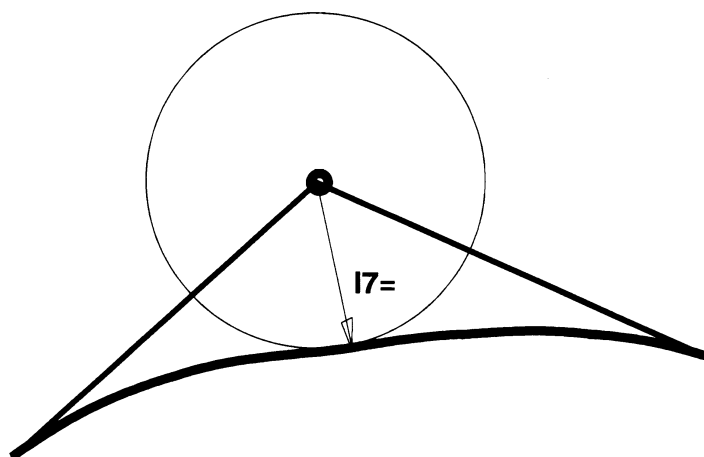
G28 I6=...

(5 to 100 %)

G28 I6=0

Setting back to normal value (100 %)

Programmable contour accuracy (rapid and feed)



NB9801b

Definition of contour accuracy

G28 I7=..

The permissible contour accuracy, in mm.

G28

When carrying out the feed or rapid movement, the contour accuracy programmed with I7=.. must be taken into account.

The feed is automatically reduced by the CNC to the maximum feed at which the corner can be executed. The programmed difference is not exceeded.

If I7= is not programmed, the numerical value established via a machine constant (MC765) is used as maximum difference.

Notes:

1. The programmable contour accuracy cannot be used for corners connected with splines.
2. The input I7=0 is turned off (MC765 is taken into account).

Cancellation parameters

CANCELLATION ALL PARAMETERS

All G28 parameters are reset to their default values by programming G27 or by performing the CLEAR CONTROL operation or softkey CANCEL PROGRAMM or M30.

G27 results in G28 I3=0 I4=0 I5=0 I6=0 I7=0

CANCELLATION OF EACH INDIVIDUAL PARAMETER

Each parameter of the G28 function can be cancelled separately by programming the default setting. The parameters do not influence each other.

Overview:

- | | |
|--|------------------------|
| 1. G28 without parameter
G1, G2, G3 with In-Position | G28 |
| 2. Movement with feed
G2, G3 without In-Position (initial setting)
G1, G2, G3 with In-Position | G28 I3=0
G28 I3=1 |
| 3. Rapid traverse movements G0
G0 with In-Position (initial setting)
G0 without In-Position | G28 I4=0
G28 I4=1 |
| 4. Positioning logic with G0
G0 with positioning logic (initial setting)
G0 with positioning logic | G28 I5=0
G28 I5=1 |
| 5. Acceleration reduction
G0, G1, G2, G3.
-Acceleration factor per axes (MC3*04)
-Acceleration reduction
I6= 5 to 100 % | G28 I6=0
G28 I6=... |
| 6. Movements with programmable contour accuracy
G0, G1, G2, G3
-Contour accuracy (MC765)
-Programmable contour accuracy
I7=... (0-10.000 µm) | G28 I7=... |

5.18 G29 Conditional jump

Purpose

To jump to a different section of a partprogram (or subprogram) if a parameter is >0.

Other jump conditions like =, <>, >, >=, <, <=, can be programmed when using a relational expression together with the G29-function.

Format

N... G29 E... N=... {K...} {I...}

Parameters

```
G    Conditional jump
I    Search direction
K    Jump decrement
E    Jump condition: E > 0
E*** Parameter definition
N=   Jump to blocknumber
```

Modal words

E...= Parameter definition.

Associated functions

G14, G22

Type of function

Non-modal.

Notes and usage

BLOCK NUMBER FOR JUMP (N=)

This word specifies the block number of the block to jump to. The block must be in the same partprogram or subprogram.

When there are block numbers with the same number then the first following block with the number is jumped to or a error is shown.

JUMP DIRECTION

A jump can be performed in forward or backward direction in a (sub) program. With I=1 or I=0 the jump is performed forwards. If I=-1 or nothing the jump is performed backwards to the top of the program and the forwards.

JUMP CONDITION (E)

The value of the E-word is used for dictating if a conditional jump must be performed.
 If the value of E...>0, a jump is performed.
 If the value of E...<=0, the jump is not performed.

JUMP DECREMENT (K)

The K-word allows the value of the E-word to be recalculated and used as the condition for jumping.

If the K-word is not programmed, the E-word's value is reduced by 1 every time a jump is made.

If K0 is programmed, the value of the E-word is not reduced.

If K>0, its value is used to reduce the E-word's value. If eg. K5 were programmed, 5 would be subtracted from the E-word's value after every jump.

If K<-.5 an error message is displayed.

DEFINING E-PARAMETERS IN A G29-BLOCK

In a G29-block parameters can be defined and calculated. The sequence of executing the block is:

1. load the parameters
2. perform the jump, if the condition is fulfilled

UNCONDITIONAL JUMP

An unconditional jump can be programmed by setting the parameter for the jump condition equal 1.

Eg. G29 E50 N=... E50=1

RELATIONAL EXPRESSIONS

When using relational expressions (see the appendix on E-parameters at the end of this manual), the programming facilities of the conditional jump are substantially expanded.

The relational expression sets the parameter for the jump condition to 0 or 1. The jump is executed as usual.

To keep a program as readable as possible, it is advised to program the relational expression in the same block as the G29. However, the relational expression can be programmed also in a block before the G29, as only the set parameter for the jump condition is used by the G29.

Eg. N.. G29 E1=E2>E3 E1 N=400

This block means:

If the value of E2 is greater than the value of E3, parameter E1 is set =1 and on this setting the jump to N400 is performed.

Example

```
:  
N50 E2=3  
N51  
:  
N100 G29 E2 N=51
```

Explanation:

N50: Set initial value of the parameter E2 to 3.

N100: When $E2 > 0$, jump to block number 51 and then continue to execute the program blocks in sequential order till block N100.

At each jump parameter E2 is decremented automatically by 1. Therefore, after 3 loops, parameter E2 is equal to zero and no more jumps executed. The program continues after block N100 in the sequential order.

5.19 G36/G37 Activate/ Deactivate turning mode

Purpose

G36 Activate turning mode. Switching on machine to endless rotating of the C-axis.

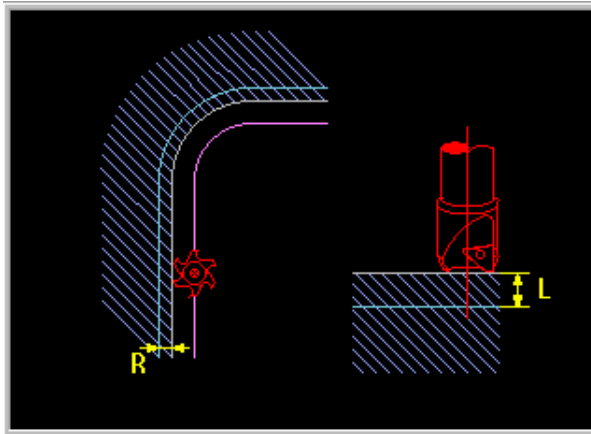
G37 Deactivate turning mode. Switching off machine to milling mode.

For explanation refer to Paragraph 12.3 in Chapter "Turning mode" on page 555

5.20 G39 Activate/Deactivate tool offset

Purpose

Programmed contours can be changed by an offset.



Format

Activate offset:

N... G39 {R...} {L...}

R: Tool radius offset

L: Tool length offset

Deactivate tool length offset:

N... G39 L0

Deactivate tool radius offset:

N... G39 R0

Parameters

G	Activate tool offset
L	Toollength offset
R	Toolradius offset

Associated functions

G17, G18, G19, G40, G41, G42, G43, G44, G61, G62, G66, G67, G87-G89, G141, G182, G201, M6, M66, M67

Type of function

Modal

Notes and usage**TOOL LENGTH OFFSET:**

The tool length offset operates into the direction of the tool axis. Tool length offset changes will become effective with the next feed movement.

TOOL RADIUS OFFSET:

The tool radius offset operates in the machining plane, but is only effective with active cutter radius compensation.

If the cutter radius compensation is inactive, tool radius offset changes will become effective after the cutter radius compensation (G41/G42, G43/G44) has been activated.

If the cutter radius compensation is active, tool radius offset changes will be corrected in the next movement block linearly over the entire path.

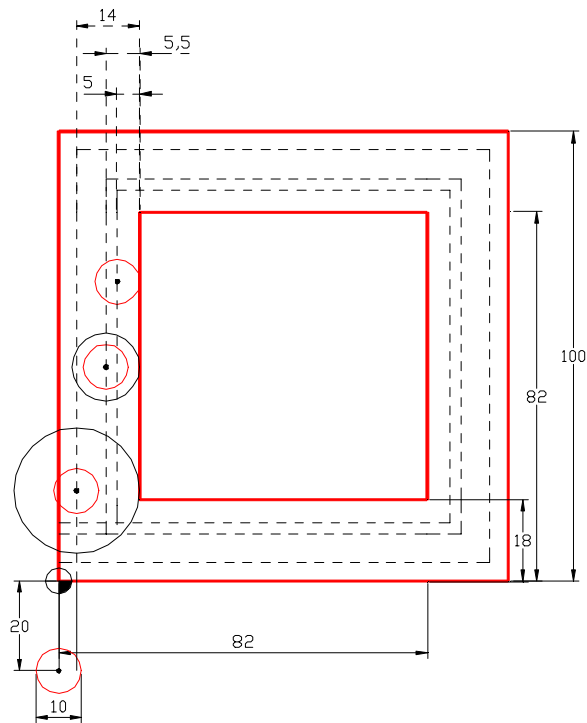
Offset programming is maintained after a tool change (M6, M66) or change of plane (G17, G18, G19).

Note:

There will be a radius offset override when the following functions are activated: G6, G83-G89, G141, G182. The length offset remains effective. Offset programming should be deactivated before these functions.

Example

Rectangular milling by roughing (2x) and finishing (1x)



```

N39001
N1 G98 X-10 Y-10 Z10 I120 J120 K-60
N2 G99 X0 Y0 Z0 I100 J100 K-40
N3 T1 M6
N4 G39 L0 R9
N5 F500 S1000 M3
N6 G0 X0 Y-20 Z5
N7 G1 Z-10
N8 G43 X18
N9 G41 Y82
N10 X82
N11 Y18
N12 X0
N13 G40
N14 G39 R0.5
N15 G14 N1=8 N2=13
N16 G39 R0
N17 G14 N1=8 N2=13
N18 G0 Z10
N19 M30

```

Explanation:

N1 Define graphic window
 N2 Define material
 N3 Change tool (cutter diameter 10 mm)
 N4 Activate tool radius offset. (Cutter radius for radius compensation is $(5+9) = 14$ mm)
 N5 Activate feed and spindle speed
 N6 Approach starting position
 N7 Moving to depth

N8 Approach contour with radius compensation
N9 Initial roughing of the rectangle. Offset is 9 mm.
N13 Turn off radius compensation.
N14 Change tool radius offset. (Cutter radius for radius compensation is $(5+0.5 =) 5.5$ mm)
N15 Repeat rectangle (second roughing operation). Offset for finishing is 0.5 mm
N16 Change tool radius offset. (Cutter radius for radius compensation is 5 mm)
N17 Finish the rectangle.
N18 Approach clearance distance
N19 Program end.

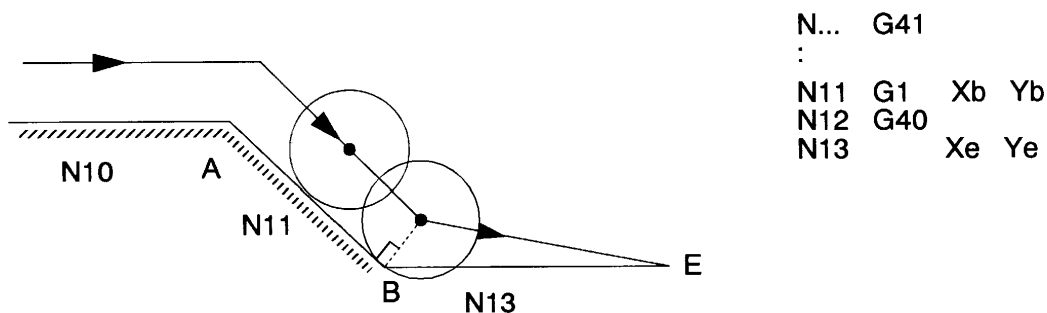
5.21 G40 Cancel tool radius compensation

Purpose

To cancel radius compensation. The tool now moves along the programmed path on the workpiece.

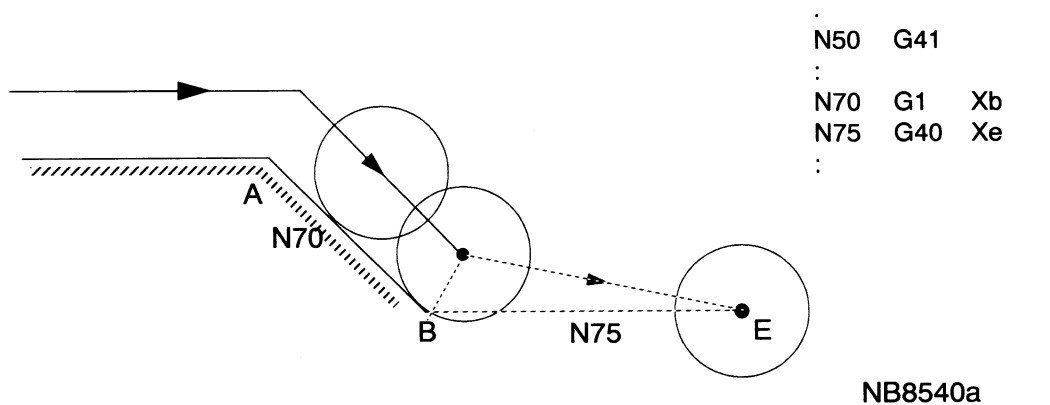
Format

N... G40 {axis coordinates}



N... G40 {without movement}

The radius compensation LEFT is active from point A to point B. At point B, radius compensation is cancelled and programmed movements refer to the tool point.



N... G40 {with movement}

Radius compensation LEFT is active from point A to point B. At point B, radius compensation is cancelled and programmed movements refer to the tool point.

Parameter

None

Associated Functions

G41/G42, G43/G44, G61/G62, G141

Type of function

Modal: (G41 to G44)

Notes and usage**DEFAULT MODE**

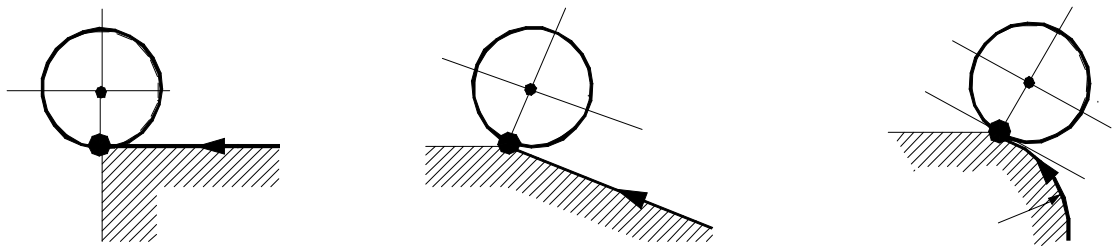
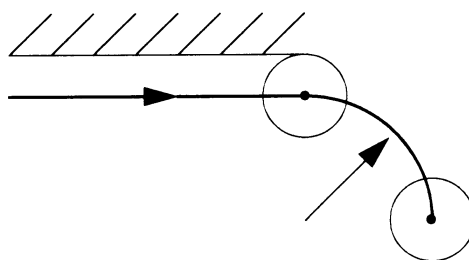
G40 is made active automatically when the CNC system is switched on, M30 and softkey CLEAR CONTROL and CANCEL PROGRAM operations are performed.

AXES COORDINATES

It is advised to program G40 in a separate block without axis coordinates.

(a) G40 without axis coordinates.

In this case, the previous line or circle will be cut completely. The centre of the tool will be moved to a position perpendicular to the contour in the previous endpoint. The compensation is switch off in the next movement.

**G40 followed by a linear movement**

```

N... G41
:
:
N... X...
N... G40
N... G2 X... Y... R...

```

NB9659

G40 followed by a circular movement

If a circular movement follows the G40-block, an arc with the programmed radius is inserted between the corrected endpoint of the G40-block and the programmed tooltip position in the block with the circle

(b) G40 with axis coordinates.

The start point of the movement is calculated with full compensation. During the movement the compensation is switched off and the endpoint is without compensation. This way of programming can be used when the compensation can be switched off without damaging a contour.

(c) Tangential exit.

It is also possible to leave a contour with G62.

N.. G62 X.. Z.. R.. (F)

LIFTING THE TOOL FROM THE PLANE OF OPERATION

The tool can be lifted from the plane of operation with

(a) A linear movement

If the plane of operation is e.g. the XY-plane and programming a linear movement in YZ lifts the tool, the tool correction in the third axis (X) is cancelled too. So a movement takes place in three axes simultaneously.

(b) A circular movement

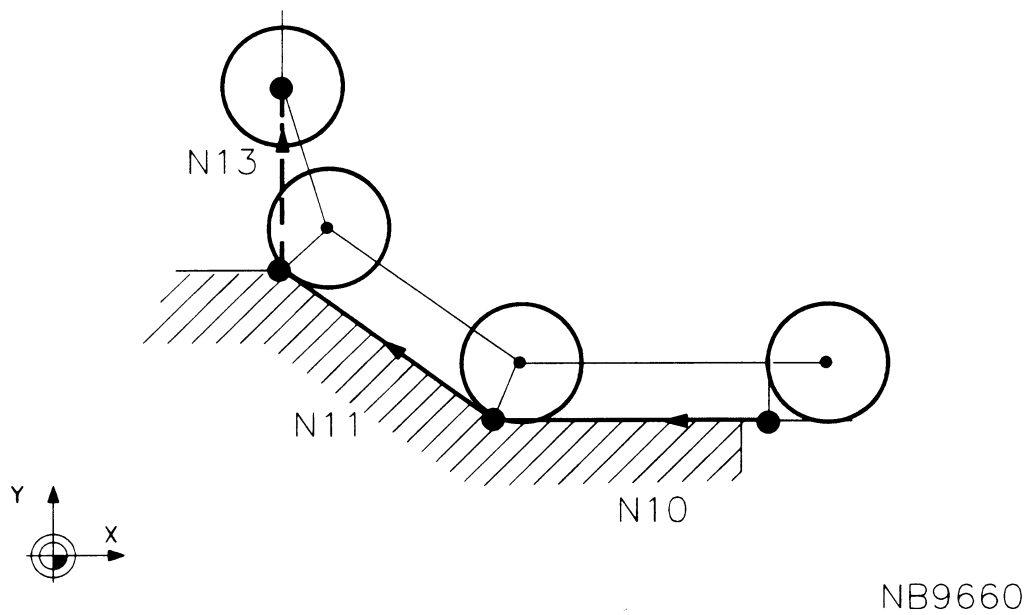
If the plane of operation is e.g. the XY-plane and programming a circular movement in YZ lifts the tool, the tool correction in the third axis (X) is not cancelled. So only the circular movement in the YZ-plane is executed.

Note: The start position of the circle is in the Y-axis the corrected position and in the tool axis (Z) the depth of operation.

Programming sub sequential identical axes positions in G43 and G40 may cause a positioning error in the axis concerned over the toolradius. This can be prevented by programming in G40 an axis position slightly different (e.g. 1 micron) from the previous position in G43.

Example: N100 G43 X10

Exampel



N9 G42
 N10 G1 X..
 N11 X... Y..
 N12 G40
 N13 G0 Y...

Explanation:

N9: activate radius compensation on right side of contour
 N10-11: move tool to programmed coordinates. Include tool radius into calculations.
 N12: cancel radius compensation.
 N13: move tool from the previous compensated position to the uncompensated endpoint of this rapid movement.

5.22 G41/G42 Tool radius compensation (left/right)

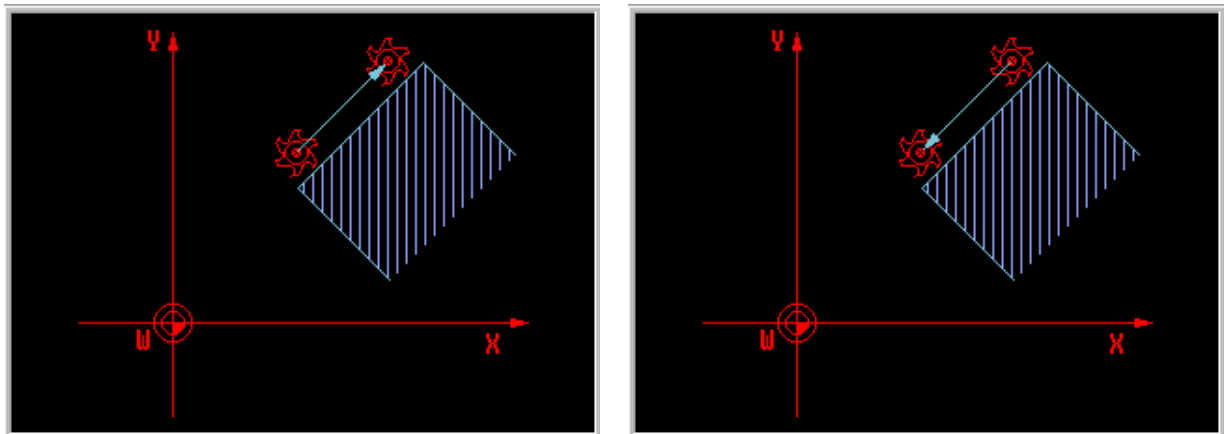
Purpose

To allow for workpiece dimensions to be programmed rather than the toolpath. The toolpath is automatically calculated by the CNC to be a path parallel to the programmed workpiece contour.

G41 activates radius compensation LEFT of the workpiece

G42 activates radius compensation RIGHT of the workpiece

In both cases when looking in the same direction as the movements of the cutting tool.



Formats

N.. G41/G42 {axis coordinates}

Parameter

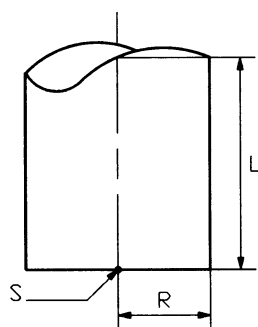
None

Associated functions

G40, G43/G44, G61/G62 and for 3D-tool correction G141

Type of function

Modal

Notes and usage**POSITION OF TOOL POINT**

NB9646

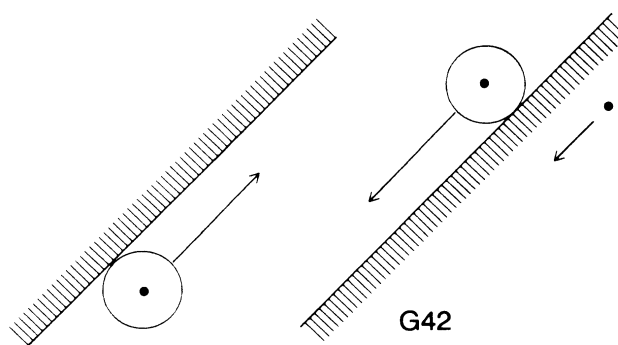
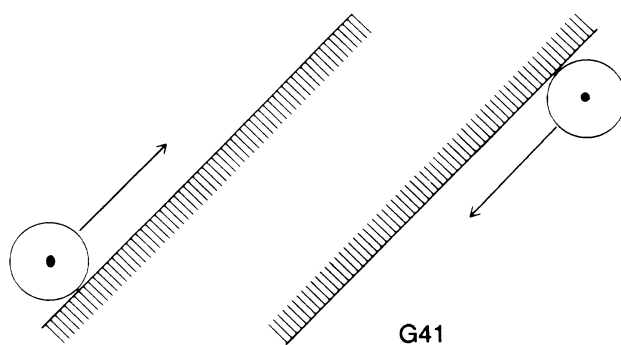
S = tool point specified by the tool dimensions L and R

TOOL MEMORY

The radius compensation function uses the tool radius from the tool memory.

COMPENSATION LEFT AND RIGHT

When using radius compensation the CNC must know whether the tool is cutting at the left or right side of the workpiece. The function G41 or G42 is used for this purpose.



NB9648

To decide which function must be programmed, it is necessary to look in the same direction as the movement of the cutting tool. If the tool moves on the left of the workpiece surface, G41 is used and on the right, G42.

This method assumes that a positive radius value is stored in the tool memory, when the program is executed.

However, if the stored radius value is negative, the following applies:

G41 and negative radius = G42 and positive radius

G42 and negative radius = G41 and positive radius

Refer to TOOL RADIUS CORRECTION for using negative radius values in the tool memory.

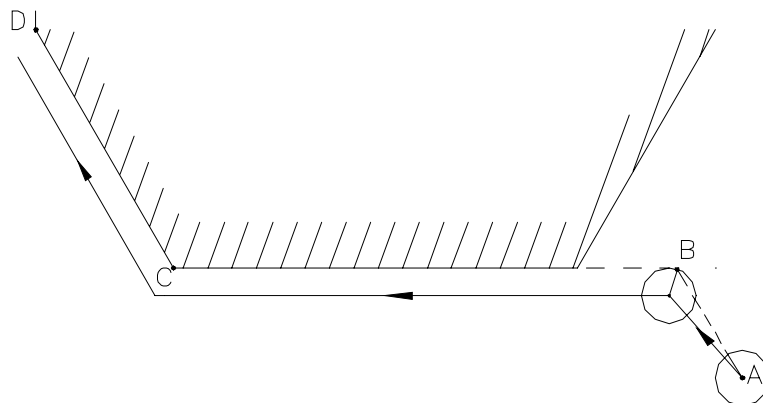
STARTING RADIUS COMPENSATION

There are three options to start radius compensations:

- 1) Direct with the functions G41/G42.
- 2) With the functions G43/G44. (See G43)
- 3) Or the function tangential approach. (See G61)

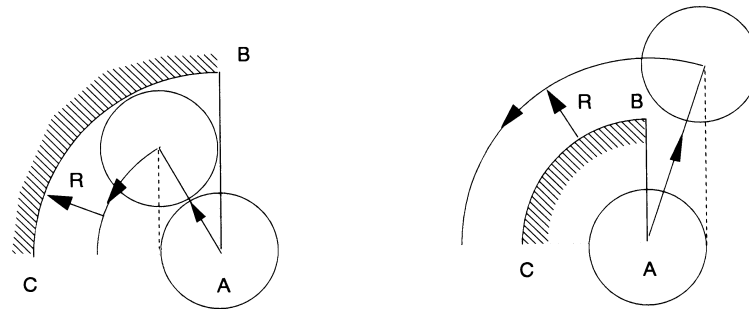
The partprogrammer must ensure that the tool does not collide with the workpiece, when radius compensation is being started. The start point must therefore be at a safe distance outside the workpiece.

When G41 or G42 is used, the intersection point between two related contour elements is calculated by the CNC and the tool moves to this point.



---Programmed path

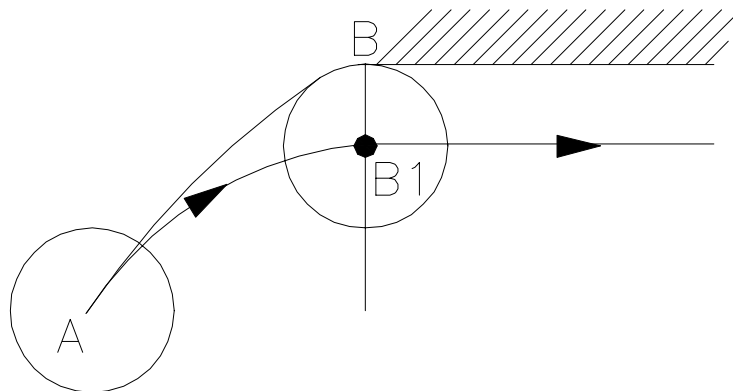
Activate radius compensation G41/G42 with line to line.



NB9650

Activate radius compensation G41/G42 with line to circle.

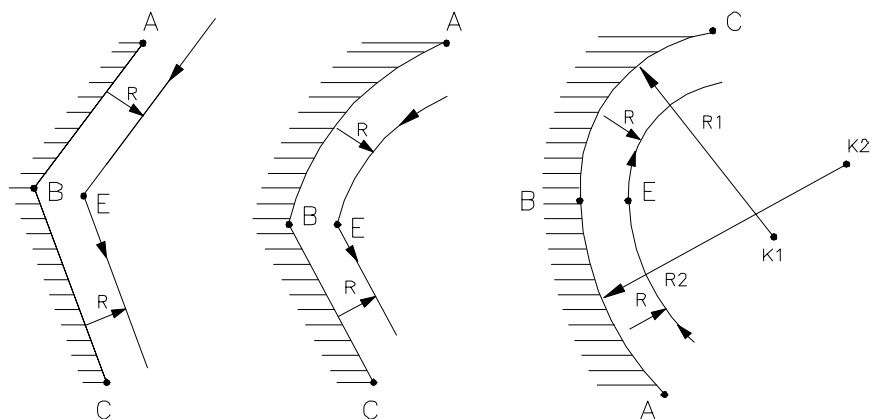
If radius compensation is activated during a circular movement (AB), the tool moves with a circular arc from the point the tool is standing (A) to the first calculated position (B1).



Activate radius compensation G41/G42 on a circle

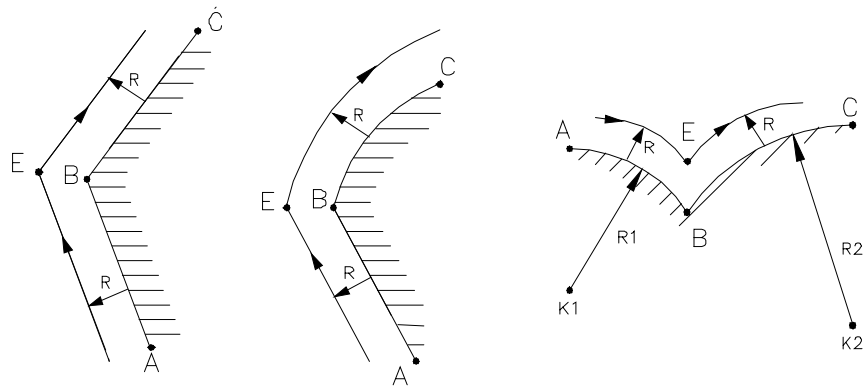
INTERNAL CONTOURS

When radius compensation is used, the toolpath is always the same distance from the programmed contour, except at intersection points between contour elements. These points are calculated by the control automatically.



EXTERNAL CONTOURS

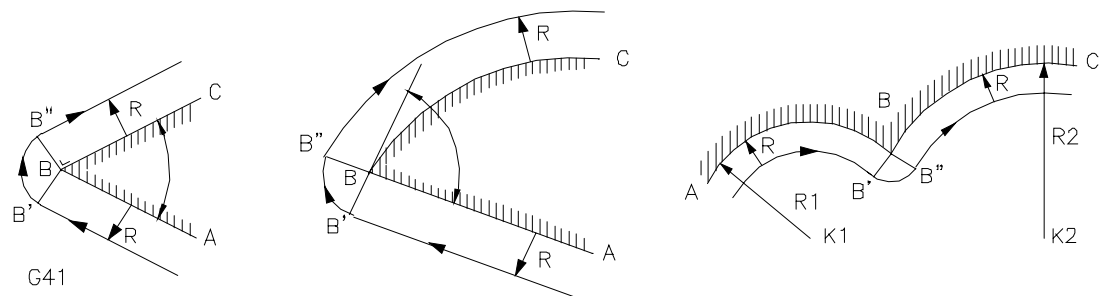
Intersection points between external contour elements are calculated and the tool moves to that position, whenever the angle between the elements is greater than a Machine Constant Value.



EXTERNAL CONTOURS WITH SHARP CORNERS

If the angle between two external contour elements is less than the Machine Constant Value (MC711), a circular movement between the two elements is generated by the CNC.

This circular movement is treated as part of the previous block. Therefore, if a SINGLE BLOCK operation is commanded, the tool stops after this circular movement.



TOOL RADIUS CORRECTION

In general a NC programming system calculates the toolpath taking into account the radius of a nominal tool. The radius compensation as described, allows to use a real tool for machining the part and to use a deviation on the radius of the nominal tool to let the control calculate the path of the actual tool.

A correction value on the tool radius including a sign is therefore stored in the tool memory.

"+" Correction value": for an oversized cutter, thus with a radius greater than the radius of the nominal tool.

"-" Correction value": for an undersized cutter

MOVEMENTS

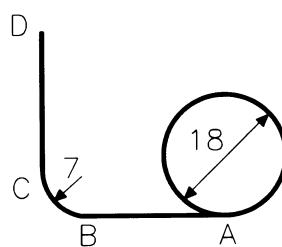
Programming either G41 or G42 does not result in a preparatory function for a motion (G0, G1, G2 or G3) also being activated. The last programmed function for a movement remains active.

PROGRAMMING ERRORS

If the tool radius is too large, the workpiece might be damaged in some situations.

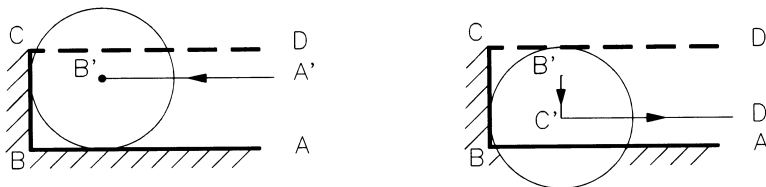
- a The radius of the tool is equal to or larger than the workpiece radius. If this occurs an error message is generated.

Note: the tool radius must be at least 0.001 mm (0.0001") smaller than the programmed radius.



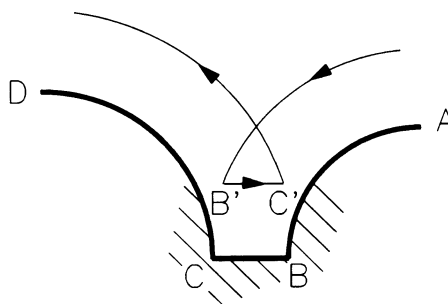
NB6250a

- b The contour from AB to BC is programmed. With active radius compensation the tool retracted along CD. If BC is smaller than two times the tool radius, the tool collides with the workpiece during the movement from B' to C' and from C' to D'.



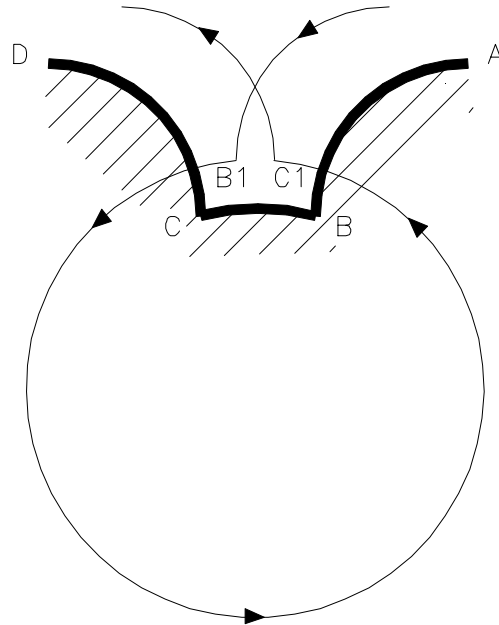
NB9620

- c A contour of the shape given in the illustration below is programmed. If the straight line is smaller than two times the tool radius, the tool collides with the workpiece during machining.



NB6250c

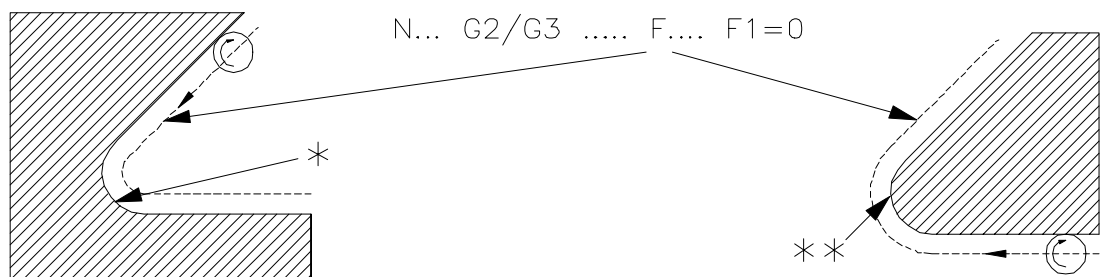
- d. A contour of the shape given in the illustration below is programmed. The tool moves to point B1, then from B1 to C1 and then parallel along CD. The movement from B1 to C1 takes place in the same direction as programmed on the circle BC. If the circular movement BC is too small, this results in the tool making almost a complete circle before it arrives at C1.



CONSTANT CUTTING FEED

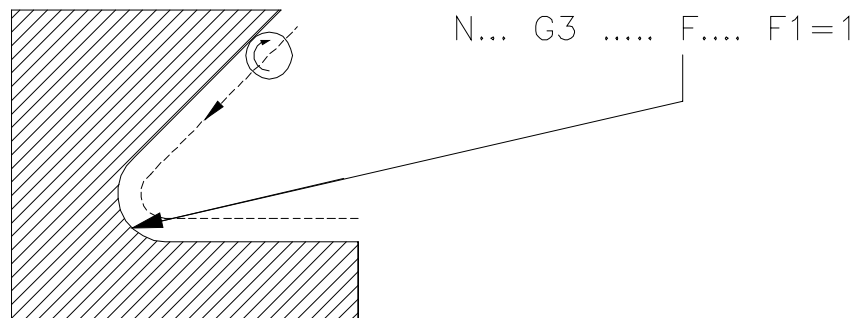
The parameter 'F1=.' is used to ensure that the programmed feed rate along a workpiece contour remains constant regardless of the radius of the mill and the contour shape. This controlled velocity is called the **CONSTANT CUTTING FEED**.

- F1=0 Constant cutting feed not applied (default mode; also set at CLEAR CONTROL or M30 or Softkey CANCEL PROGRAM). The programmed feed rate should be the velocity of the tooltip.



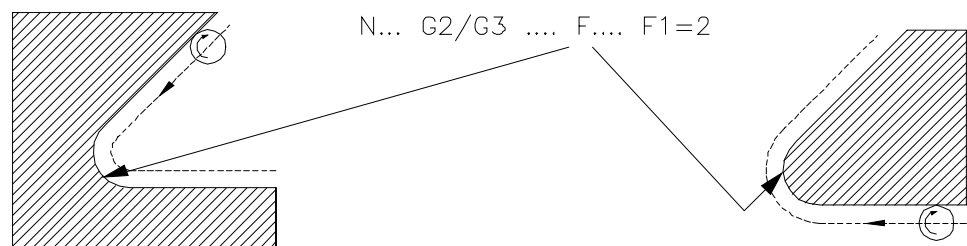
- * Cutting feed too high
 ** Cutting feed too low

- F1=1** Constant cutting feed applied only on the inside of arcs. The programmed feed rate is reduced to assure that the tooltip moves with the reduced velocity on the inside of an arc.

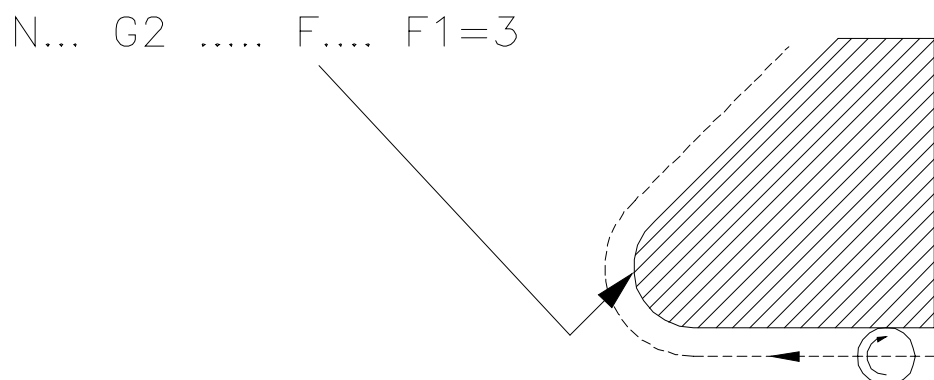


NF

- F1=2** Constant cutting feed applied on the inside and outside of arcs. The programmed feed rate is reduced (inside arc) or increased (outside arc) to assure that the tooltip moves with the recalculated velocity. If the increased velocity is greater than the maximum feed rate (a Machine Constant value) the maximum feed rate is used.



- F1=3** Constant cutting feed applied only on the outside of arcs. The programmed feed rate is increased to assure that the tooltip moves with the increased velocity on the outside of an area. If the increased velocity is greater than the maximum feed rate (a Machine Constant value) the maximum feed rate is used.



SWITCHING FROM ONE RADIUS COMP. FUNCTION TO ANOTHER ONE

When switching from one function, e.g. G41 to G42, G43 or G44, the tool ends in a position, which is calculated with the first function active and starts in a position calculated with the other function active. When these two positions do not coincide a linear feed movement from one position to the other one is executed.

ENDING RADIUS COMPENSATION

The function G40 cancels the tool radius compensation. Thereafter programmed coordinates refer to movements of the tooltip.

PLANE FOR RADIUS COMPENSATION

The radius compensation is performed in the plane indicated by G17, G18 and G19.

TOOL AXIS MOVEMENT

A simultaneous movement of the tool axis and the axes of the main plane (defined with G17, G18 or G19) with activated radius compensation is possible. The compensated movements in the main axes can be linear or circular.

HELIX INTERPOLATION

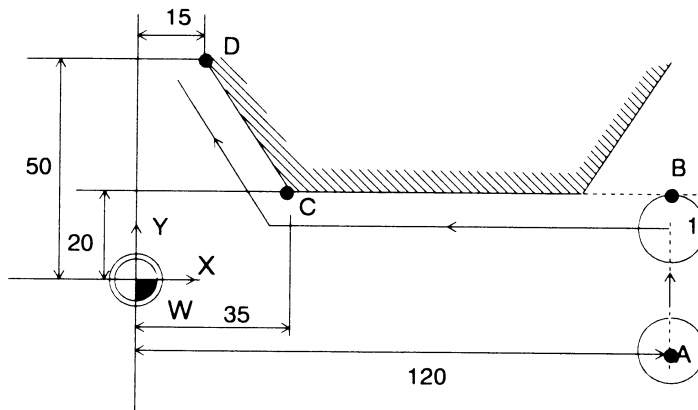
If helix interpolation is used, radius compensation can be used on the circular movement in the plane indicated with G17, G18 or G19.

USING THE CYLINDRICAL COORDINATE SYSTEM

When the cylindrical coordinate system (G182) is activated, the functions G41/G42/G43/G44 as described can also be used in the plane of the cylinder.

Examples

EXAMPLE 1.



NB8604

```

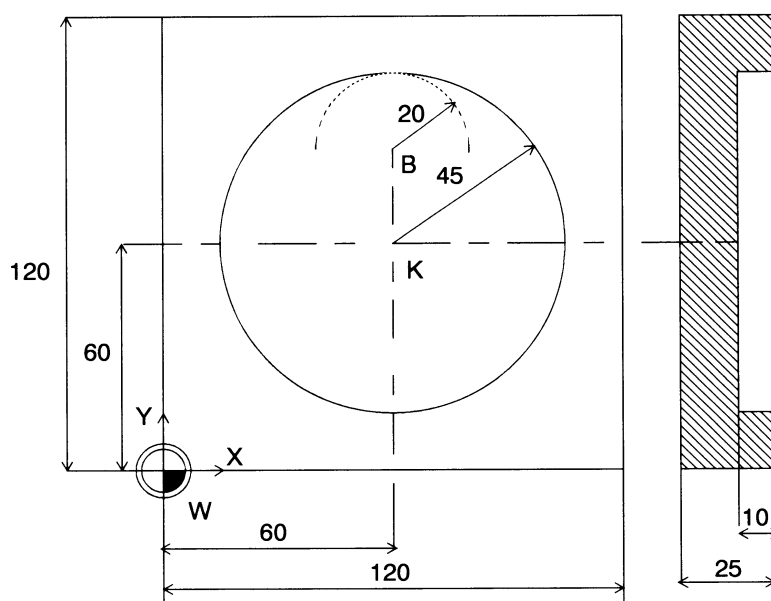
N9999
N1 G17
N2 G54
N3 T1 M6
N4 G0 X120 Y-20 Z5 S3000 M3
N5 G1 Z-10 F600
N6 G43
N7 Y20
N8 G41
N9 X35
N10 X15 Y50
N11 G40
N12 G0 X120 Y-200 Z5 M30

```

Explanation:

N3:	Load tool 1 and its offsets. Mill diameter is 10 mm
N4:	Start spindle, move the tool rapidly to point A
N5:	Moving to depth with a feedrate of 600 mm/min.
N6:	Set radius compensation to.
N7:	Move the tool to point B.
N8-N10:	Select radius compensation RIGHT. Move the tool at set feed rate Along the right hand side of the workpiece
N11:	Cancel radius compensation.
N12:	Move tool rapidly to point A.

EXAMPLE 2.



NB8603

```

N9998
N1 G17
N2 G54
N3 T1 M6
N4 G0 X60 Y85 Z0 S3000 M3
N5 G1 Z-10 F500
N6 G43 X80 F300
N7 G41
N8 G3 X60 Y105 R20
N9 I60 J60
N10 X40 Y85 R20
N11 G40
N12 G0 Z200 M30

```

Explanation:

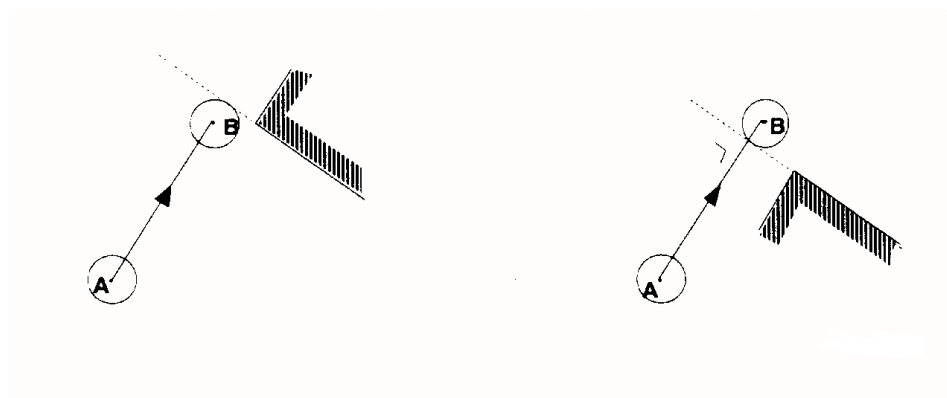
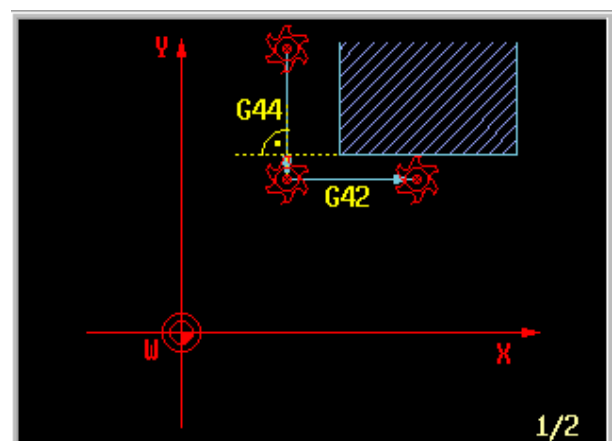
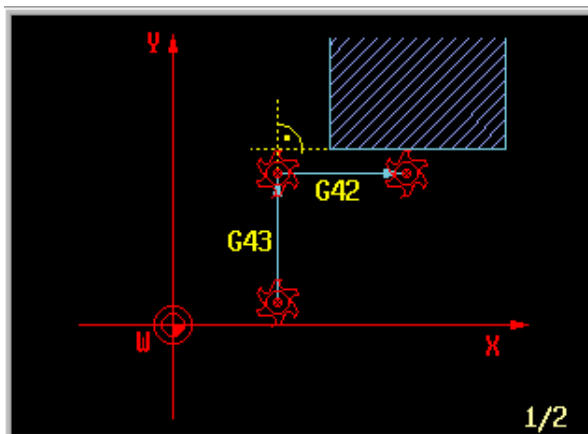
N3: Load the tool. The mill has a diameter of 10 mm.
 N4: Start the spindle and move tool to starting point B.
 N5: Feed the tool to depth.
 N6: Move the tool to the starting point of the small circle. Set new feed rate to 300 mm/min.
 N7: Set radius compensation LEFT.
 N8: Move the tool with a circular movement, to enter the contour.
 N9: Mill the complete circle.
 N10: Exit the contour by using a small circular movement.
 N11: Cancel the radius compensation.
 N12: Retract the tool out of the workpiece. End of program.

5.23 G43/G44 Tool radius compensation to/past endpoint

Purpose

To move the tool with cutter radius compensation. TO/PAST a programmed position.

- G43 activates radius compensation 'TO' a programmed position (tool radius is subtracted from the programmed position).
 G44 activates radius compensation 'PAST' a programmed position (tool radius is added to the programmed position).



G43 'TO'

G44 'PAST'

Format

N... G43/G44 {axis coordinates}

Parameter

None

Type of function

Modal

Associated functions

G40, G41/G42, G61/G62 and for 3D-tool correction G141

Alternative

G43 is mostly used for axis parallel positioning movements. If the positioning movement is not parallel to the axis, starting point B should be calculated. It is therefore wiser to use the possibilities of G61 (tangential approach).

Notes and usage

CIRCULAR MOVEMENT

If a G43 or G44 is active with a circular movement (G2/G3), an error message is displayed. With circular movements G41 or G42 must be used.

AXIS PARALLEL MOVEMENT

If a G43 or G44 is used with an axis parallel movement and only one coordinate is programmed, the position in that axis is calculated. The other axis remains unchanged. So G43/G44 only can be used with radius compensation and axis parallel movements.

ENTERING A CONTOUR WITH G43 OR G44

The functions G43 and G44 can be used to enter a contour on the normal (= perpendicular to) of any contour element. This way of entering a contour is recommended because there is less change of accidental collision between tool and workpiece than the direct use of the functions G41 or G42. See also STARTING RADIUS COMPENSATION with G41/G42.

SWITCHING FROM ONE RADIUS COMP. FUNCTION TO ANOTHER ONE

When switching from one function, e.g. G41 to G42, G43 or G44, the tool ends in a position, which is calculated with the first function active and starts in a position calculated with the other function active. When these two positions do not coincide a linear feed movement from one position to the other one is executed.

ENDING RADIUS COMPENSATION

The function G40 cancels the tool radius compensation. Thereafter programmed coordinates refer to movements of the tooltip.

PLANE FOR RADIUS COMPENSATION

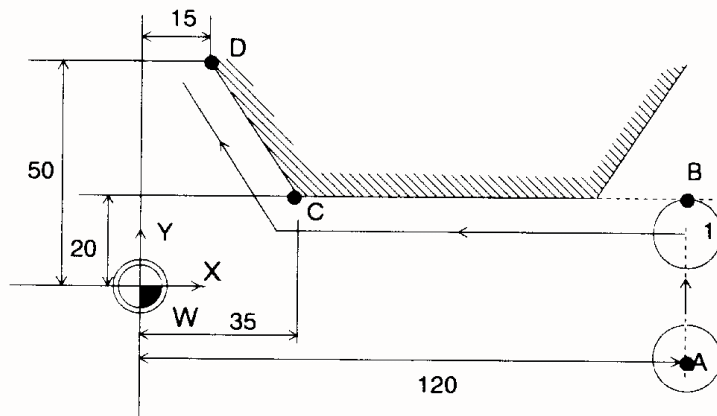
The radius compensation is performed in the plane indicated by G17, G18 and G19.

USING THE CYLINDRICAL COORDINATE SYSTEM

When the cylindrical coordinate system (G182) is activated, the functions G41/G42/G43/G44 as described can also be used in the plane of the cylinder.

Examples

EXAMPLE 1.



```

N40 G0 X120 Y-15 Z10
N41 G1 Z-10 F500
N42 G43 Y20
N43 G41 X35
N44 X15 Y50

```

Explanation:

N40: Move the tool to the starting point A and feed to depth.
 N41: Set the feed rate to 500 mm/min.
 N42: Move the tool to the workpiece.
 N43: Start first milling movement along the left hand side of the contour.

Alternative with G61

```

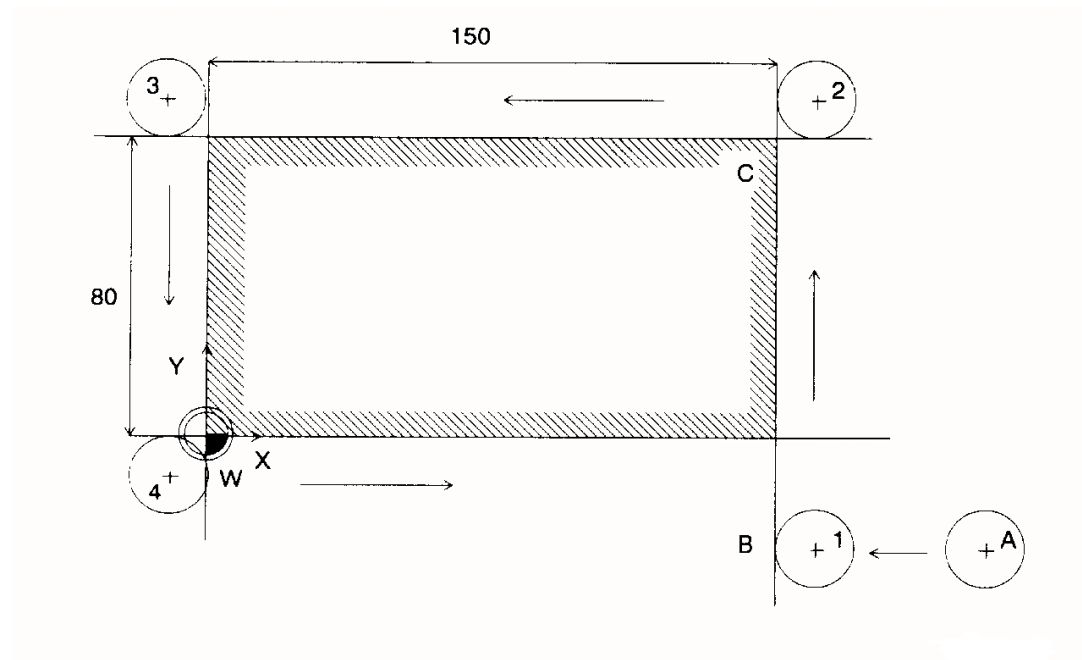
N41 G41
N42 G61 X120 Y20 Z-10 Z1=10 I2=5 R20 F500
N43 G1 X35
N44 X15 Y50

```

Explanation:

N41: Next milling movement will be along the left hand side of the contour.
 N42: Move the tool with a vertical approach to the starting point (Starting point Z1=...)
 Feed rate 500 mm/min
 X, Y, Z: Starting point of the contour
 Z1: Starting point Z
 I2=5: vertical
 R: Distance between the vertical and the start point.
 N43: Start first milling movement along the left hand side of the contour.

EXAMPLE 2.



```

N9 T1 M6
N10 G0 X200 Y-20 Z-5 S1000 M3
N11 G43 X150
N12 G1 F200
N13 G44 Y80
N14 X0
N15 Y0
N16 X150
N17 G40 Y-20
N18 G0 X200

```

Explanation:

- N9: Load tool 1 and its offsets.
 N10: Make spindle rotate clockwise at 1000 rev/min, move the tool to position A and then at depth.
 N11: Move the tool rapidly to point B.
 N12: Set linear feed rate to 200 mm/min.
 N13: Move the tool along the Y-axis PAST edge Y80 (point 2). The function G44 remains active in the blocks that follow.
 N14: Move the tool along the X-axis PAST edge X0 (point 3).
 N15: Move the tool along the Y-axis PAST edge Y0 (point 4).
 N16: Move the tool along the X-axis PAST edge X150. The tool is free from the part
 N17: Cancel the radius compensation.
 N18: Rapid traverse movement to position A.

5.24 G45 Axis parallel measuring movement and measuring tool dimensions

Note

Use of this function is limited only to programs made on earlier control systems.

Two actions can be done with G45:

- 1) Measuring a point with only G45
- 2) Measuring tool dimension G45 + M25

5.24.1 G45 Measuring a point

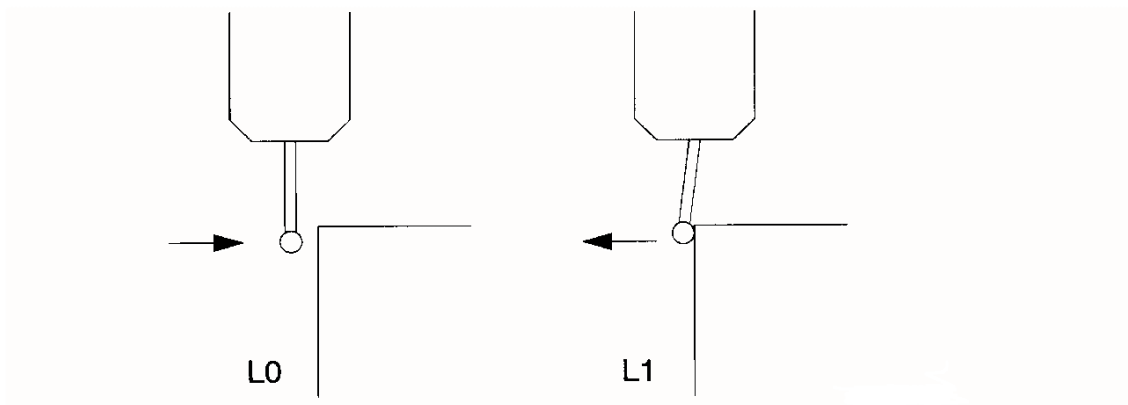
Note

Use of this function is limited only to programs made on earlier control systems.

The G45 function operates only parallel to the axis. G145 has improved functionality and is also able to perform measurements, which are not parallel to the axis. It is therefore wiser to use the new G145 basic measurement movement.

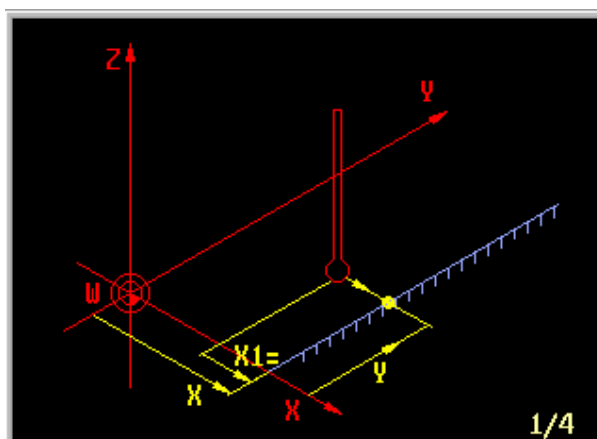
Purpose

To measure the actual axis coordinate with a touch trigger probe when being moved in the axis to the programmed position. This allows the difference between the actual and programmed position to be used to check the dimensional accuracy of the workpiece.



Probe moves towards the workpiece

Probe is 'triggered'



Format

N... G45 [Measuring position] {I+/-1} {J+/-1} {K+/-1} {L+/-1} {X1=...} {N=...} {E...}

The plane for the rotary table is determined by the definition of the 4th axis in the machine constant list. (MC117 should be 4 and MC118 should be B (66) or C (67)). L relates to the 4th axis B or C. Rotary axis A is not allowed.

Parameters

G Measuring a point
 X Measurement target coordinate
 Y Measurement target coordinate
 Z Measurement target coordinate
 A Measurement target angle
 B Measurement point angle
 I Measurement direction for X axis
 J Measurement direction for Y axis
 K Measurement direction for Z axis
 L Measurement direction rotary-axis
 M M25 for tool measurement
 E Parameter-nr measured coordinate
 N= Point-nr.for measured coordinate
 X1= Measurement path length

?90= Measurement target abs. (X,Y,Z..)
 ?91= Measurement target incr.(X,Y,Z..)
 P1= Point definition number

Measuring position

X, Y, Z Measurement target coordinate
 C Measurement target angle
 P Point definition number

Measuring parameters

I Measurement direction for X axis
 J Measurement direction for Y-axis
 K Measurement direction for Z-axis
 L Measurement direction rotary-axis
 X1= Measurement path length

Measuring results

E Parameter number measured coordinate
 N Point number for measured coordinate

The difference between the measured and programmed coordinate is calculated and stored internally for use with G49 or G50.

Modal words

E...=

Type of function

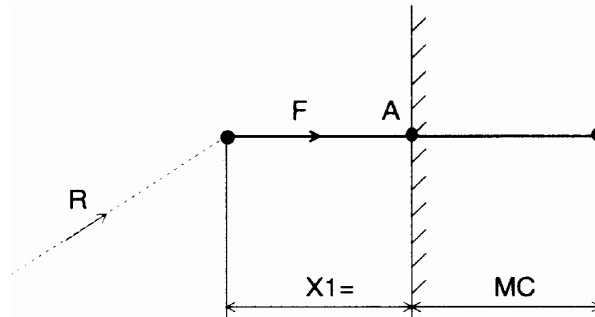
Non-modal.

Associated functions

G46, G49, G50 and the basic measuring movement G145
 M25, M27, M28

Notes and usage**MEASURING POSITION**

The programmed coordinates specify the point to be measured.

PRE- AND POST-MEASURING DISTANCE (X1=)

The pre-measuring distance defines the position in the axis to be measured from where the movement with the measuring feed starts. This distance is programmed with the word X1=.

If X1= is not programmed, a Machine Constant value (MC844) is used.

With the post-measuring distance (MC845) is defined how far the probe can pass the programmed position of the axis before it is triggered.

MEASURING SEQUENCE

1. The probe moves rapidly to the pre-measuring position, which is defined, by the programmed position and the pre-measuring distance in the axis to be measured. This movement is executed with the positioning logic of G0.
2. After the probe reaches the pre-measuring point it moves at a fixed feed rate (MC843) along the indicated axis in the programmed direction towards the programmed position in the axis. The probe can pass this point, but must be triggered along the path between the pre- and post-measuring distance.
3. When the probe touches the workpiece, the measured coordinate is stored and the probe rapidly moves back to the pre-measuring position.

STORING MEASURING RESULT (E, N=)

The measured coordinate can be stored in either the E-Parameter Memory (E) or and the Point Memory (N=).

Storing the coordinate in an E-parameter has the advantage of allowing additional calculations to be performed, such as in a macro.

The difference between the measured and programmed coordinate is calculated and stored internally for use with G49 or G50.

The stored differences are cancelled as soon as a new measuring function (G45 or G46) is activated or with Softkey CLEAR CONTROL or CANCEL PROGRAM.

ERROR MESSAGES

An error message is displayed and the movement stops,

1. If the probe touches an obstruction during the rapid movement to the pre-measuring position,
2. If the probe exceeds the post-measuring distance.

COLLISION PROTECTION

As soon as the measuring probe is triggered during any other movement than the actual feed movement for the measurement it, an error is generated and the movement interrupted. Sometimes the probe is triggered due to very fast movements and not by a real collision.

A machine constant (MC850) can be used to store the information that collision protection

- Is switched off during the measurement movement and possibly during retraction after measurement;
- Is effective during all movements or only during feed movements.

TOOL MEMORY

The radius of the probe and its length are stored in the tool memory together with a tool number. Tool type Q3=9999 can be entered to indicate the measuring probe.

Example: P5 T5 Q3=9999 L150 R4

When tool T5 is called with Q3=9999 the control system recognizes this tool as the measuring probe. The probe radius is called and used to correct the measurement position.

If a function for spindle direction (M3 or M4) is entered, this function is suppressed and an error message displayed.

AIR BLOW BEFORE MEASURING

To clean the workpiece at the position to be measured an air blow can be executed during a fixed time (MC842).

The air blow is activated once by an M-function and executed each time a pre-measuring position is reached. Refer to the machine tool builder's documentation for the number of the M-function for the air blow.

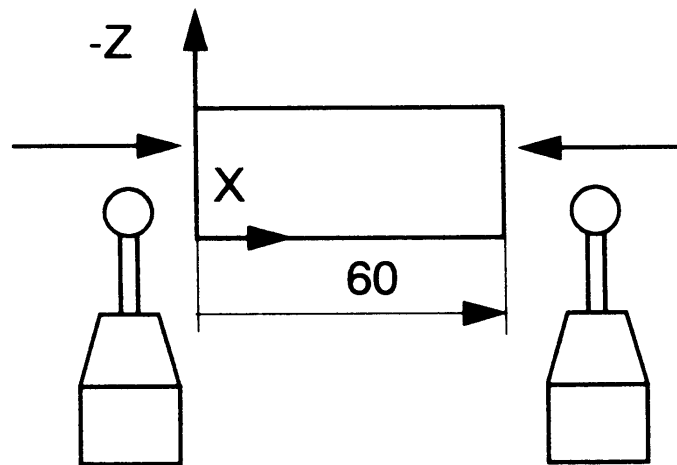
RESTRICTION

1. Only one axis coordinate can be measured in a G45-block.
2. An omni directional probe should be used.
3. In the tool axis the probe is only triggered if pressed. This means that a measurement in the positive direction of the tool axis is not possible.

Note: The G45 function is also used in conjunction with the M25 function for measuring tool dimensions.
Refer to G45 + M25 section for additional information.

Example

EXAMPLE 1: Measuring a point in X-axis



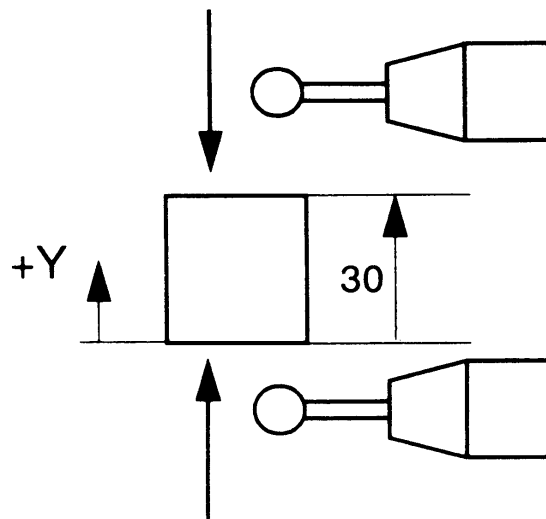
Measuring in positive direction

N.. G45 X0 Y20 Z-10 I1 E1 N=1

Measuring in negative direction

N.. G45 X60 Y20 Z-10 I-1 E1 N=1

EXAMPLE 2: Measuring a point in Y-axis



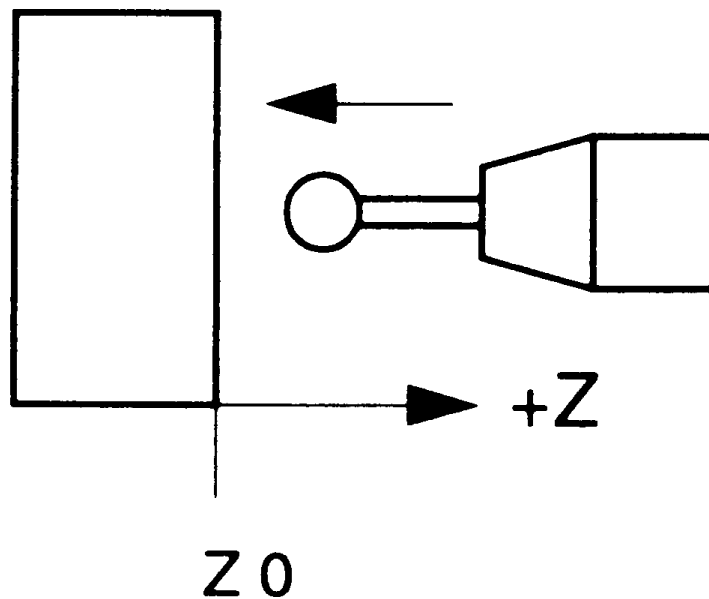
Measuring in positive direction

N.. G45 X30 Y0 Z-10 J1 E1 N=1

Measuring in negative direction

N.. G45 X30 Y30 Z-10 J-1 E1 N=1

EXAMPLE 3: Measuring a point in Z-axis



Measuring in negative direction

N.. G45 X30 Y30 Z0 K-1 E1 N=1

Note: Measuring in the tool axis is only possible in the negative direction.

Explanation of the examples:

The point is measured; the measured position calculated and stored in Point Memory location 1 and parameter E1.

5.24.2 G45 + M25 Measure tool dimensions

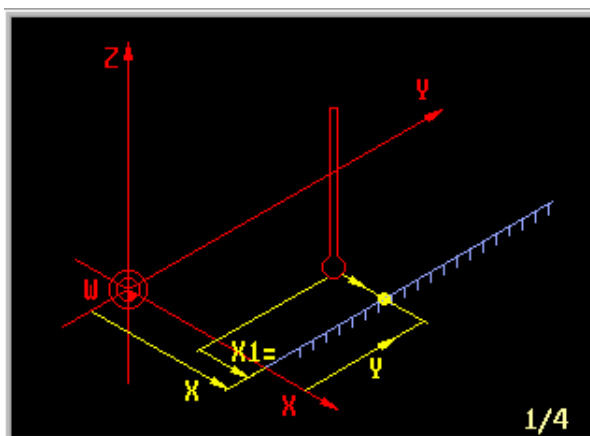
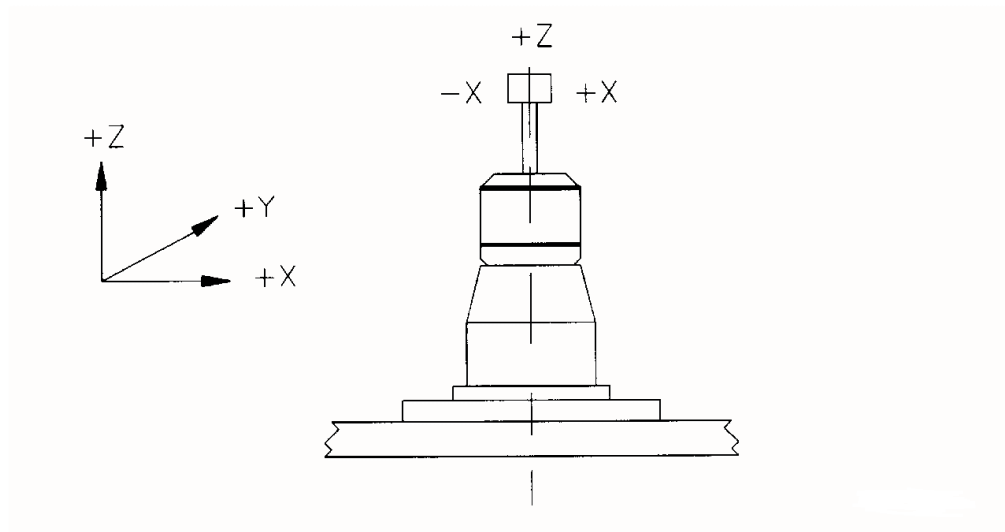
Note

Use of this function is limited only to programs made on earlier control systems.

The G45 function operates only parallel to the axis. G145 has improved functionality and is also able to perform measurements, which are not parallel to the axis. It is therefore wiser to use the new G145 basic measurement movement.

Purpose

To measure tool dimensions using an square-head measuring probe.



Format

N... G45 {I...} {J...} {K...} {X1=...} M25

Parameter

G Measuring a point
 X Measurement target coordinate
 Y Measurement target coordinate
 Z Measurement target coordinate
 A Measurement target angle
 B Measurement point angle
 I Measurement direction for X axis
 J Measurement direction for Y axis
 K Measurement direction for Z axis
 L Measurement direction rotary-axis
 M M25 for tool measurement
 E Parameter-nr measured coordinate
 N= Point-nr.for measured coordinate
 X1= Measurement path length

?90= Measurement target abs. (X,Y,Z..)

?91= Measurement target incr.(X,Y,Z..)

P1= Point definition number

I Measurement direction for X axis
 J Measurement direction for Y-axis
 K Measurement direction for Z-axis
 L Measurement direction rotary-axis
 X1= Measurement path length

Modal words

E...=

Type of function

Non-modal.

Associated functions

G45, G46, G49, G50
 M26, M27, M28

Notes and Usage**MEASURING TOOL DIMENSIONS**

An square-head-measuring probe mounted at a fixed position on the machine tool, is used for measuring the tool dimensions.

Measuring in the tool axis gives the tool length.

Measuring in two directions of the same axis gives the tool radius.

POSITION OF THE SQUARE-HEAD PROBE

The position of the square-head measuring probe (MC3155, MC3?55) and its width are red in the Machine Constant Memory (MC847).

MEASURING SEQUENCE

The measurements are executed in the same way as with G45. Instead of programming the position of the fixed probe, its coordinates are picked up from the Machine Constant Memory.

UPDATING THE TOOL MEMORY

The tool memory is updated with the function G50. Refer to that function for details.

Note

1. Tool measurement can also be performed on the control in the mode OPERATE. Refer to the Operating Manual for details.
2. Refer to G145 for an example for automatic measurement of the tool dimensions with a measuring box.

Example

Measuring tool length

```
N89 T1 M6  
N90 G45 K-1 X1=5 M25  
N91 G50 T1 L1=1
```

Explanation:

N89: Load tool 1
N90: Measure the tool length in the negative direction of the Z-axis. Pre-measurement distance is 5 mm.
N91: Correct the tool length of tool 1 in the tool memory

5.25 G46 Measuring a full circle or probe calibration

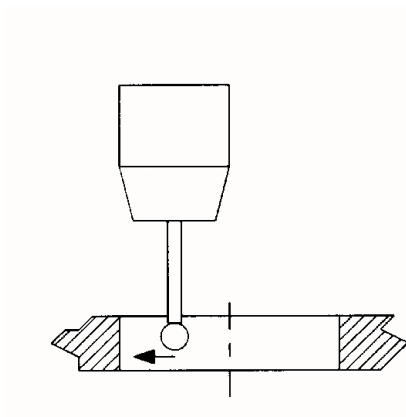
Two actions can be done with G46:

- 1) G46 Measuring a full circle with only G46
- 2) Probe calibration with G46 + M26

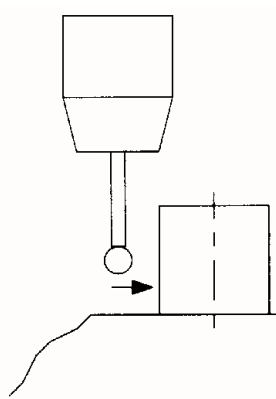
5.25.1 G46 Measuring a full circle

Purpose

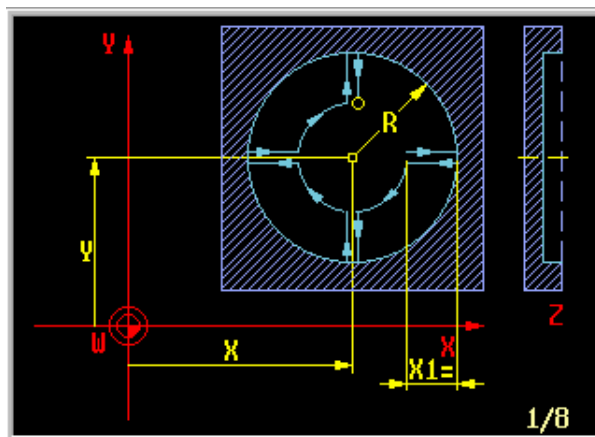
To measure a full circle and to determine the centre point coordinates and any deviation between the programmed circle radius and the calculated radius.



Measuring an inner circle



measuring an outer circle



Format

Measuring an inner circle

N... G46 [Circle centre coordinates.] R... {1+1 J+1} {1+1 K+1} {J+1 K+1} {F...} {X1=...} N=... E...

Measuring an outer circle

N... G46 [Circle centre coordinates.] R... {1-1 J-1} {1-1 K-1} {J-1 K-1} {F...} {X1=...} N=... E...

Parameters

G Measuring a circle
X Center point coordinate
Y Center point coordinate
Z Center point coordinate
A Measurement target angle
B Measurement target angle
I Measurement direction for X axis
J Measurement direction for Y axis
R Circle radius
M M26: probe radius measurement
E Parameter-nr. measured radius
N= Point-nr.measured centre point
X1= Measurement path length
?90= Centre point abs. (X,Y,Z..)

?91= Centre point incr. (X,Y,Z..
P1= Point definition number

Circle parameters

X, Y, Z Centre point coordinate
 C Measurement target angle
 P Point definition number
 R Circle radius

Measuring parameters

I Measurement direction for X axis
 J Measurement direction for Y-axis
 K Measurement direction for Z-axis
 F Feed between measurements
 X1= Measurement path length

Measuring results

E Parameter number measured radius
 N= Point number measured centre point

Modal words

E...=

Type of function

Non-modal.

Associated functions

G45, G49, G50
 M26, M27, M28

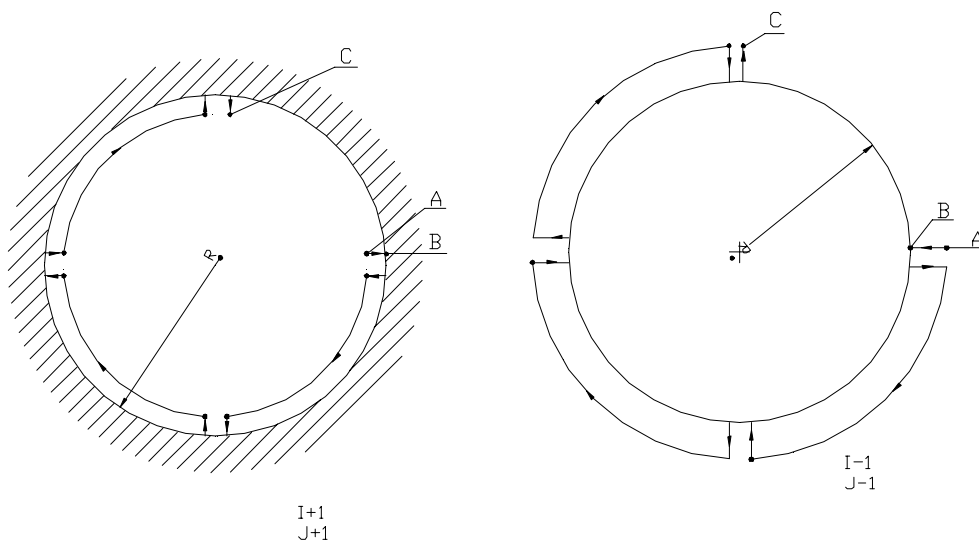
Notes and Usage

The G46 function is for measuring a full circle. Under the linear measurement movement G145 a macro is describes, with which a full circle can be measured.

MEASURING THE POSITIONS

Four positions are measured when a G46 block is executed. The measurements take place as if four G45 blocks where programmed. So refer to G45 for additional information about PRE- AND POST- MEASURING DISTANCE, TOOL MEMORY, AIR BLOW and COLLISION PROTECTION.

MEASURING SEQUENCE



A= Start point (is positioned with rapid speed)

B= Measurement point

C= End point

1. The probe moves rapidly to the pre-measuring position of the first point to be measured. This position is defined by the programmed circle centre, the programmed radius and the pre-measuring distance ($X1=$). This movement is executed with the positioning logic of G0.
2. The probe moves at a fixed feed rate (MC 843) towards the first point on the programmed circle. The probe can pass the point, however, it must be triggered along the path between the pre- (MC 844) and post-measuring (MC 845) distance.
3. When the probe is correctly triggered, the measured position is automatically stored. Then the probe moves back (at the set feed rate) to the starting position and with the programmed feed rate (F-word) along the circle in a clockwise direction until it reaches the second pre-measuring position.
4. The procedure just given is repeated for the second, third and fourth position.
5. When the fourth position has been measured, the circle centre and radius are calculated from the four measured points. The coordinates of the circle centre are stored in the Point Memory and the radius in the E- parameter memory.

MEASURING INNER OR OUTER CIRCLE (I/J/K)

Any pair of the addresses I, J, K simultaneously define the type of circle to be measured and the plane in which the circle is located. A pair of addresses must be stated in each G46-block.

Plane	Inner Circle		Outer Circle	
XY (G17)	I+1	J+1	I-1	J-1
XZ (G18)	I+1	K+1	I-1	K-1
XZ (G19)	J+1	K+1	J-1	K-1

STORING CENTRE POINT COORDINATES (N=)

The word N= states the number in the Point Memory where the calculated coordinates of the centre point are stored. E.g. N=12, means that the centre point coordinates are stored in P12.

STORING THE CIRCLE RADIUS (E)

The E-word states the number of the E-parameter where the calculated radius is stored. E.g. E45 means that the circle radius is stored as the value of E-parameter 45.

ERROR MESSAGES

An error message is displayed and the movement stops,

1. If the probe touches an obstruction during the movement to the premeasuring position,
2. If the probe exceeds the post-measuring distance.

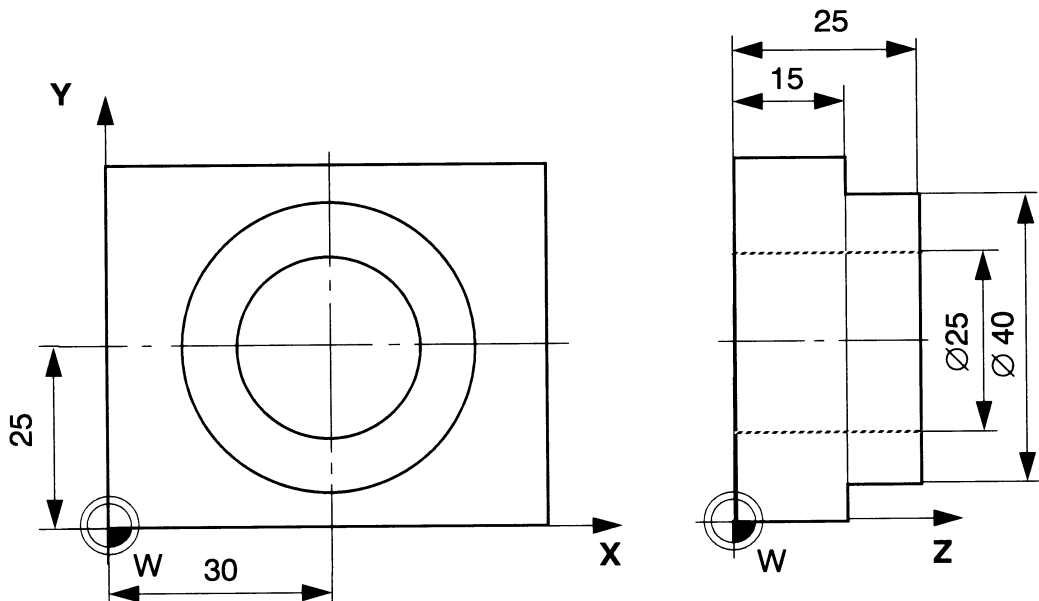
RESTRICTION

Each G46-block can measure only one circle.

Note: The G46 function is also used in conjunction with the M26 function for probe calibration. Refer to G46 + M26 section for additional information.

Example

Measuring an inner and outer circle in the XY-plane



Measuring the inner circle:

N... G46 X30 Y25 Z20 I+1 J+1 R12.5 F3000 N=59 E24

Measuring the outer circle:

N... G46 X30 Y25 Z20 I-1 J-1 R20 F3000 N=58 E23

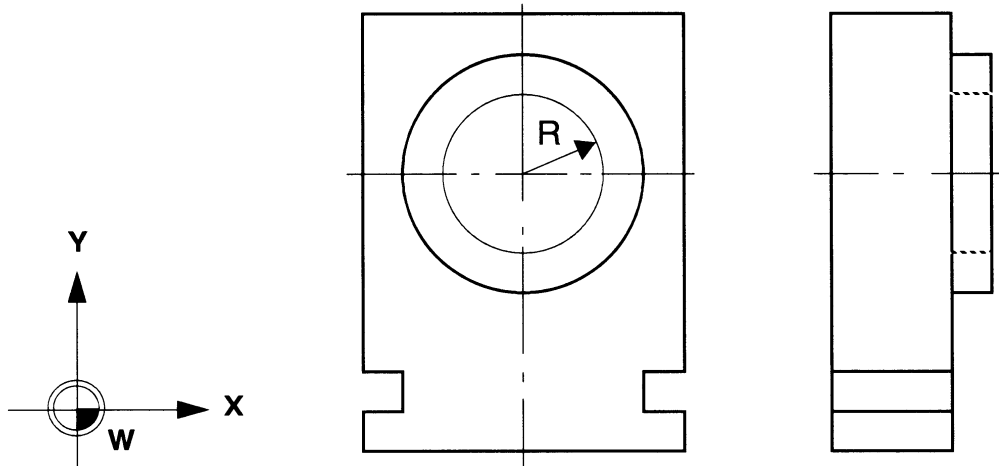
Explanation:

The circles are measured, the centre points of the measured circles calculated and stored in the Point Memory and the radius calculated and stored in the Parameter Memory.

5.25.2 G46 + M26 Probe calibration

Purpose

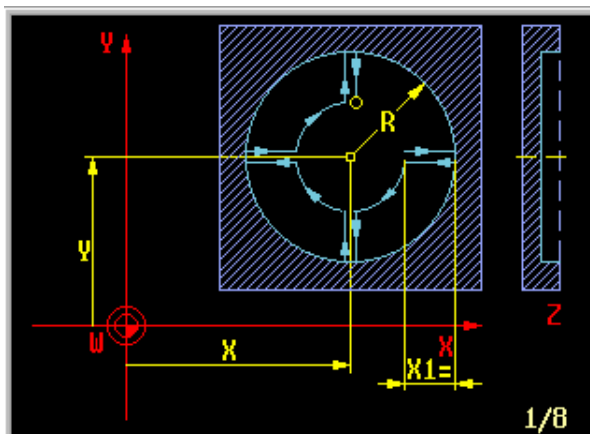
To determine the radius of a touch trigger probe by touching a calibration ring, thus a ring gauge whose diameter is exactly known.



A probe must be calibrated:

- when the probe is used for the first time
- when a new stylus is used
- after any suspected bending of the stylus.

Note: It is assumed that the position of the ball centre relative to the spindle axis is already determined.



Format

Measuring an inner ring gauge

N... G46 {I+1 J+1} {I+1 K+1} {J+1 K+1} {F...} {X1=...} M26

Measuring an outer ring gauge

N... G46 {I-1 J-1} {I-1 K-1} {J-1 K-1} {F...} {X1=...} M26

Parameters

G Measuring a circle
X Center point coordinate
Y Center point coordinate
Z Center point coordinate
A Measurement target angle
B Measurement target angle
I Measurement direction for X axis
J Measurement direction for Y axis
R Circle radius
M M26: probe radius measurement
E Parameter-nr. measured radius
N= Point-nr.measured centre point
X1= Measurement path length
?90= Centre point abs. (X,Y,Z..)

?91= Centre point incr. (X,Y,Z..
P1= Point definition number

I Measurement direction for X axis
J Measurement direction for Y-axis
K Measurement direction for Z-axis
F Feed between measurements
X1= Measurement path length

Modal words

E...=

Type of function

Non-modal

Associated functions

G45, G46, G49, G50
 M25, M27, M28

Notes and Usage**POSITION OF THE CALIBRATION RING**

The position of the calibration ring and its radius are stored in the Machine Constant Memory.

MEASURING SEQUENCE

The measurements are executed in the same way as with G46. Instead of programming the position of the ring gauge, its coordinates are picked up from the Machine Constant Memory.

MEASURING INNER OR OUTER RING GAUGE (I/J/K)

Any pair of the addresses I, J, K simultaneously define the type of ring to be measured and the plane in which the ring is located. A pair of addresses must be stated in each G46-block.

Plane	Inner ring		Outer ring	
XY (G17)	I+1	J+1	I-1	J-1
XZ (G18)	I+1	K+1	I-1	K-1
YZ (G19)	J+1	K+1	J-1	K-1

UPDATING THE RADIUS OF THE PROBE

The difference between the radius of the ring stored in the Machine Constants and the measured radius is used to update the probe radius and store this value in the tool memory for the active tool (= the probe).

WHEN PROBE CALIBRATION IS REQUIRED

A probe should be calibrated in the cases mentioned above and also:

- If accuracy demands it.
- If the repeatability of relocation of the probe in the spindle is poor. In this case calibrating may be required each time the probe is selected.

Example

```
N46002
N1 G17
N2 T1 M6
N3 D207 M19
N4 G46 I1 J1 M26
N5 Z200 M30
```

Explanation:

- N1: Set the plane of operation to be the XY-plane
- N2: Load the touch trigger probe
- N3: Stop the spindle in a defined position
- N4: Calibrate the probe by moving it to the inside surface of a ring gauge located in the XY- plane. The measured radius of the touch probe is stored in the tool memory location of the active tool (T1). A default MC value is used for the pre-measuring distance.

5.26 G49 Checking on tolerances

Note

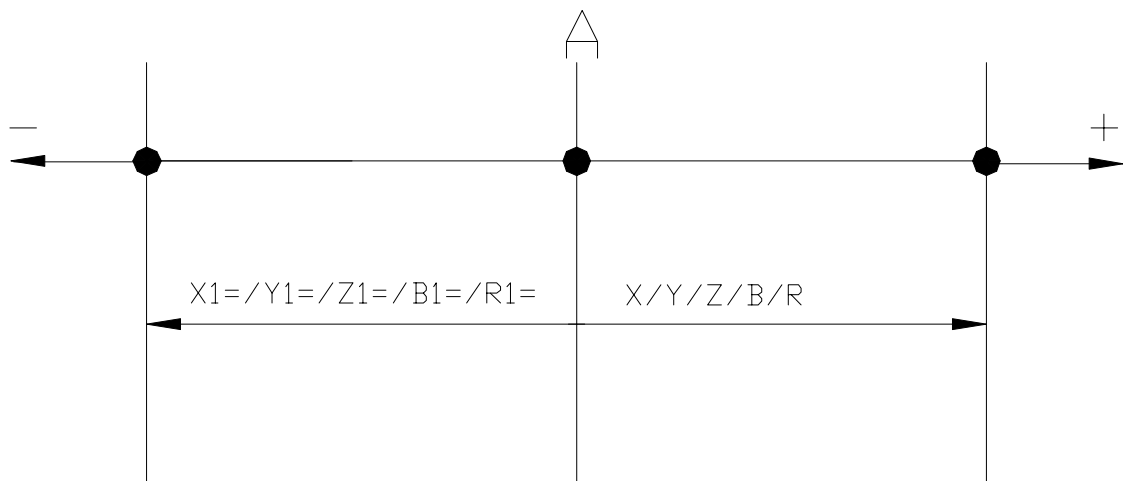
Use of this function is limited only to programs made on earlier control systems.

Since the G45 uses an internal memory, the G49 function can only be used together with G45. Basic measurement movement G145 uses E-parameters and may have the same functionality. It is therefore wiser to use the new basic measurement movement G145.

Purpose

To check whether the difference between a programmed value and the measured (G45/G46) value lies within set tolerance limits. If the difference is within the limits the program is allowed to continue. However, if the difference is not within limits there can be:

- a repeat of a section of the program until the difference is acceptable
- a conditional jump in the program
- a display of an error message.



A= Programmed point

The measured point must lie within the highest tolerance limit (X/Y/Z/B/R) and the lowest tolerance limit (X1 =/Y1 =/Z1 =/B1 =/R1 =).

Format

A repeat of a program section

N... G49 {X..., X1=...} {Y..., Y1=...} {Z..., Z1=...} {B..., B1=...} {R..., R1=...} N1=... {N2=...} {E..}

The plane for the rotary table is determined by the definition of the 4th axis in the machine constant list. (MC117 should be 4 and MC118 should be B(66) or C(67)). B, B1 relates to the 4th axis B or C. R applies to the plane of the rotary table. Rotary axis A is not allowed.

A conditional jump.

N... G49 {X..., X1=...} {Y..., Y1=...} {Z..., Z1=...} {B..., B1=...} {R..., R1=...} N=... E...

Display an error message

N... G49 {X..., X1=...} {Y..., Y1=...} {Z..., Z1=...} {B..., B1=...} {R..., R1=...}

Parameters

G Checking on tolerances
 X Positive tolerance value in X
 Y Positive tolerance value in Y
 Z Positive tolerance value in Z
 A Positive tolerance value in A
 B Positive tolerance value in B
 R Positive tolerance circle radius
 E Jump condition: $E > 0$
 N= Jump to blocknumber
 N1= Repeater begin block
 N2= Repeater end block
 X1= Negative tolerance value in X
 Y1= Negative tolerance value in Y
 Z1= Negative tolerance value in Z

A1= Negative tolerance value in A
 B1= Negative tolerance value in B
 R1= Negative tolerance circle radius

Tolerance values

X Positive tolerance value in X
 X1= Negative tolerance value in X
 Y Positive tolerance value in Y
 Y1= Negative tolerance value in Y
 Z Positive tolerance value in Z
 Z1= Negative tolerance value in Z
 B Positive tolerance value in B
 B1= Negative tolerance value in B
 R Positive tolerance circle radius
 R1= Negative tolerance circle radius

Conditional jump

E Jump condition: $E > 0$
 N= Jump to block number

Repeat of program section

N1= Repeater begin block
 N2= Repeater end block

Modal words

E...=

Type of function

Non-modal.

Associated functions

G45, G46, G50

Note

If the difference between the measured value and the programmed value is within the set tolerances, the program continues with the block after the G49 or G50.

Notes and Usage**CONTINUATION IF VALUES ARE WITHIN LIMITS**

If the difference between the measured value and the programmed value is within the set tolerances, the program continues with the block after the G49.

REPEAT OF A PROGRAM SECTION (E, N1=, N2=)

The words E, N1= and N2= are used to repeat a section of the program when a tolerance limit is exceeded.

The E-word specifies the number of repeats (E>0).

When no number of repeats is programmed (no E-word is present), the sequence is repeated only once.

BLOCK NUMBERS OF REPEAT SEQUENCE (N1=, N2=) '

These block numbers must be in the same partprogram or subprogram.

If N2= is not programmed, only the block indicated by N1= is repeated the specified number of times.

ORDER OF BLOCKS TO BE REPEATED

The order of executing the blocks in the repeat sequence must be the same as the order originally programmed. So in the program block N1=.. must be before block N2=..

CONTINUATION AFTER THE REPEAT

Once the repeats are executed, the program continues with the block after the G49.

CONDITIONAL JUMP (N=, E)

The words N= and E are used to specify a conditional jump when a tolerance value is exceeded.

The value of parameter E must be greater than zero before a jump can occur.

The word N= states the block number in the same program or subprogram to which control will jump when E>0.

CONTINUATION IF NO JUMP IS EXECUTED

If no jump is executed, because the limits are not exceeded or E<=0, program execution continues with the block after G49.

ERROR MESSAGE

An error message is generated by the CNC, if the measured value exceeds a tolerance limit and neither a repeat of a program section nor a conditional jump is programmed.

CONTINUATION AFTER AN ERROR MESSAGE

After resetting the error program execution continues with the block after G49.

CHECKING ON THE HIGHEST AND LOWEST TOLERANCE LIMIT

If tolerance checks are used to see if a part is made within tolerances, two G49-blocks can be used. The order must be:

1. Check to see if the highest tolerance limit is exceeded. If this occurs the part is too big, so a jump out of the measuring section of the program is necessary.
2. Check to see if the lowest tolerance limit is exceeded. If this occurs the part is too small and has to be milled again, so a repeat of the milling section with an updated tool radius is necessary.

Examples

EXAMPLE 1: A repeat of a program section

N97 G49 X0.005 X1=0.002 N1=80 N2=95 E2

Explanation:

If the measured position is more than 0.005 mm higher or 0.002 mm lower than the programmed position, the program section from block number N80 to N95 is repeated two times. After the repeat program execution continues from the block after N97.

Alternative:

E2 is the desired position

E3 is measured position due to G145.

N97 G29 E0 E0=E3<(E2-0.002) N=100

N98 G29 E0 E0=E3>(E2+0.005) N=100

N99 G14 N1=80 N2=95 J2

N100 ...

Explanation:

N97: Jump to N100 when measure position smaller is then 0.002 mm.

N98: Jump to N100 when measure position greater is then 0.005 mm.

N99: Repeat program twice.

EXAMPLE 2: A conditional jump

N197 G49 X0.005 X1 =0.002 E10 N=80

Explanation:

If the measured position is more than 0.005 mm higher or 0.002 mm lower than the programmed position and the value of parameter E10 is greater than zero, a jump in the program to block N80 is performed and program execution continues from that block.

Alternative:

E2 is the desired position.

E3 is the measured position due to G145.

N96 G29 E10 N=99

N97 G29 E0 E0=E3<(E2-0.002) N=80

N98 G29 E0 E0=E3>(E2+0.005) N=80

N99 ...

EXAMPLE 3: A conditional jump and repeat of program section

```
N10 G49 R.02 R1=2 E1 N=13 E1=1  
N11 G49 R2 R1=.02 N1=1 N2=6
```

Explanation:

- N10: If the measured position is more than 0.02 mm higher than the programmed position, jump to N13. R1= is set high to avoid that this limit is exceeded.
- N11: If the measured position is more than 0.02 mm lower than the programmed position, repeat the program section from N1 to N6. R is set high to avoid that this limit is exceeded.

Alternative:

E2 is the desired position of the circle radius.

E3 is measured position of the circle radius

```
N10 G29 E0 E0=E3>(E2+0.02) N=80
```

```
N98 G29 E0 E0=E3>(E2-0.02) N=95
```

```
N99 G14 N1=1 N2=16
```

5.27 G50 Processing measuring results

Note

Use of this function is limited only to programs made on earlier control systems.

Since the G45 uses an internal memory, the G50 function can only be used together with G45. Basic measurement movement G145 uses E-parameters and may have the same functionality. It is therefore wiser to use the new basic measurement movement G145.

Purpose

To make corrections derived from the measured differences on either the zero offsets or the tool dimensions.

Format

To change zero offsets

With standard zero offsets or MC84=0:

N... G50 {X1} {I...} {Y1} {J...} {Z1} {K...} [{B1}{C1}{C2}] [{B1=..}{C1=..}] {L...} N=...

With MC84>0 zero offsets extends:

N... G50 {X1} {I...} {Y1} {J...} {Z1} {K...} [{B1}{C1}{C2}] [{B1=..}{C1= }] {L...} **N=54.[nr]**

The plane for the rotary table is determined by the definition of the 4th axis in the machine constant list. (MC117 or MC120 should be 4 and the associated axes should be B(66) or C(67)). B1 or C1 relates to the 4th axis B or C. Rotary axis A is not allowed.

To change the tool length

N... G50 T... L1=1 {I...} {J...} {K...} {T2=...}

To change the tool radius.

N... G50 T... R1=1 {X1=...} {T2=...}

Parameters

G	Processing measuring results
X	X1: zero point shift in X
Y	Y1: zero point shift in Y
Z	Z1: zero point shift in Z
A	A1: zero point shift in A
B	B1: zero point shift in B
I	Multiplication factor for X
J	Multiplication factor for Y
K	Multiplication factor for Z
L	Multipl. factor for rotary-axis
T	Tool dimensions to be corrected
N=	Offset-nr for correction (52-59)
X1=	Multiply factor for tool radius
A1=	Prog.angle in A after calculation

B1=	Prog.angle in B after calculation
L1=	L1=1: correction of tool length
R1=	R1=1: correction of tool radius

Zero offsets

N=	Offset-nr for correction (52-59)
X	X1: zero point shift in X
Y	Y1: zero point shift in Y
Z	Z1: zero point shift in Z
B	B1: zero point shift in B
C	C1: zero point shift in C
I	Multiplication factor for X
J	Multiplication factor for Y
K	Multiplication factor for Z
L	Multiply factor for rotary-axis
B1=	Prog. angle in B after calculation
C1=	Prog. angle in C after calculation

Tool dimensions

T	Tool dimensions to be corrected
X1=	Multiply factor for tool radius
L1=	L1=1: correction of tool length
R1=	R1=1: correction of tool radius

Modal words

E...=, T2=

Type of function

Non-modal.

Associated functions

G45, G46, G49

Notes and Usage

TO CHANGE OFFSET VALUES (N=)

With the G50 function new offset values derived from the measured corrections can be stored in the Zero Offset Memory.

MULTIPLICATION FACTOR FOR AXES (I, J, K,)

A multiplication factor can be applied to the measured difference, e.g. K8, means multiply Z-axis difference by 0.8.

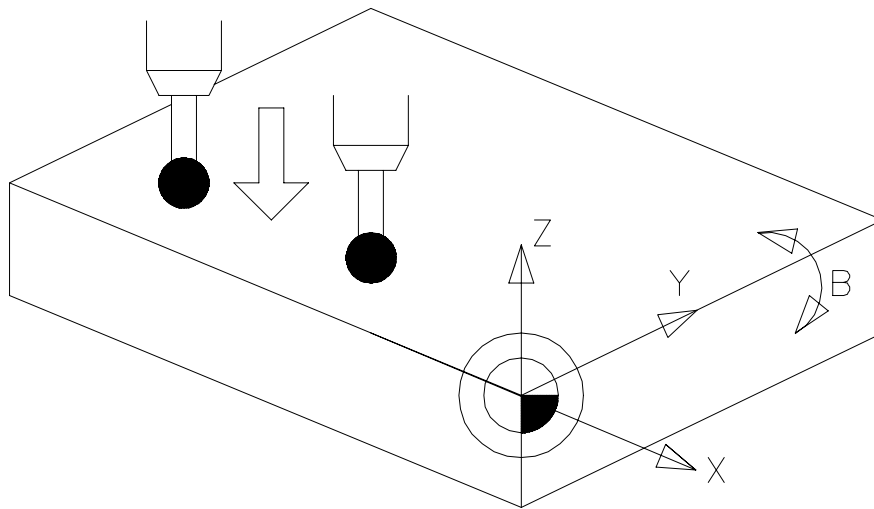
The multiplication factor can have a positive or negative value.

If no factor is stated, the default value +1 will be used automatically by the CNC.

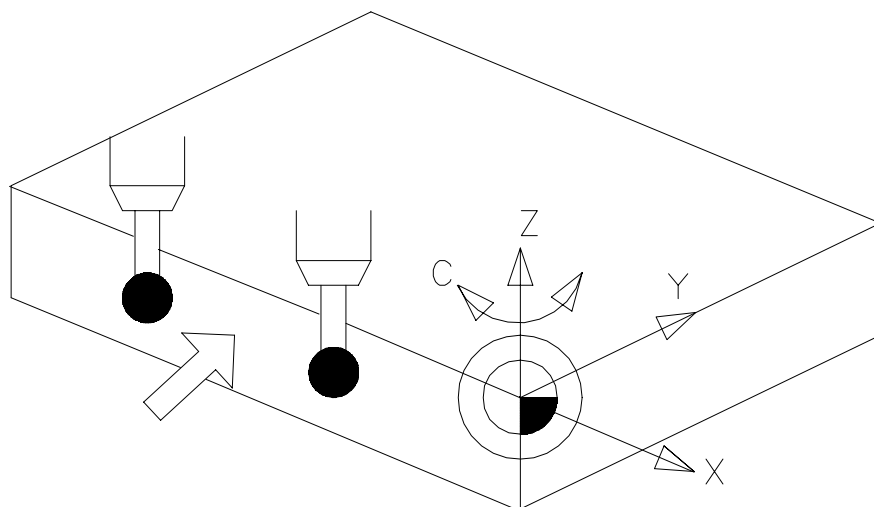
MACHINE CONFIGURATIONS (B1, C1, C2)

B-Axis B1: Aligning of a workpiece, mounted on a round table (B-axis) turning around the Y-axis, the measurement of two point in X direction are sufficient.

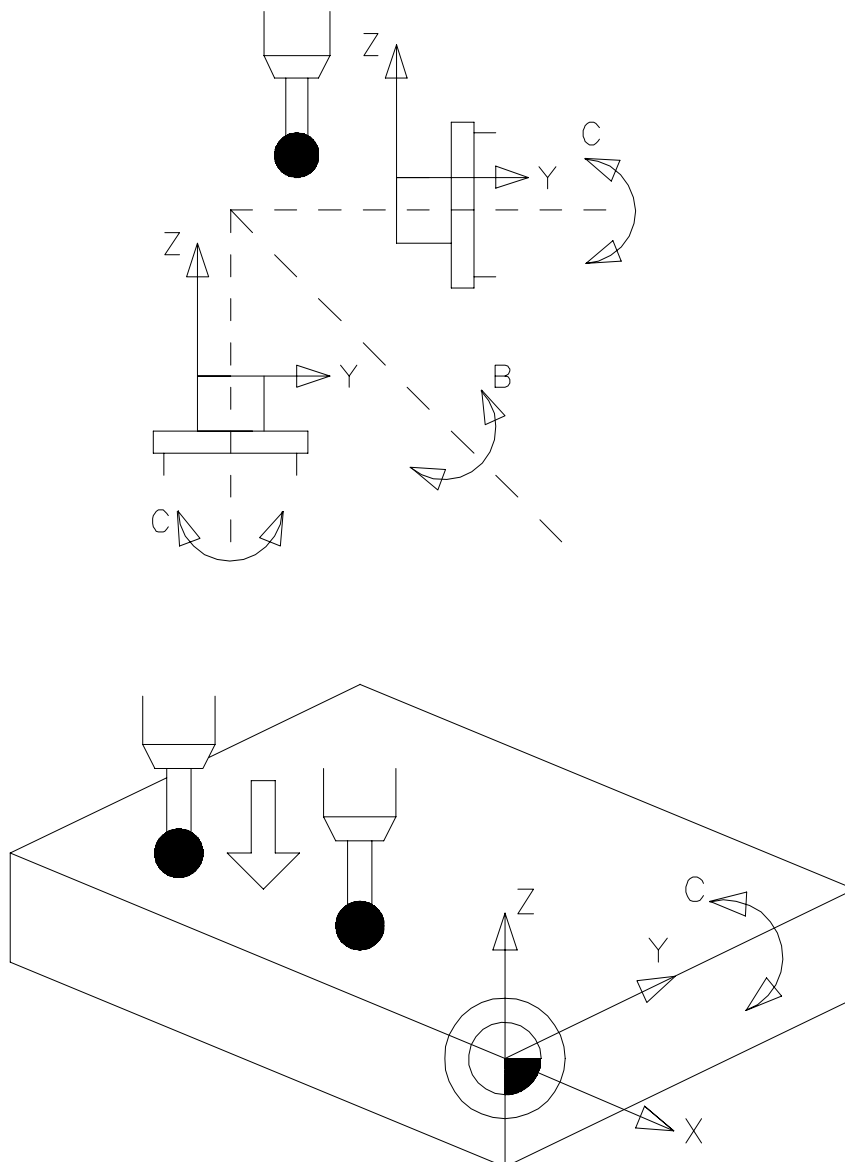
- The rotations angle is in respect with the X-axis.
- The workpiece is turning around the Y-axis.
- The measuring tracer stands in the Z-or Y-direction.



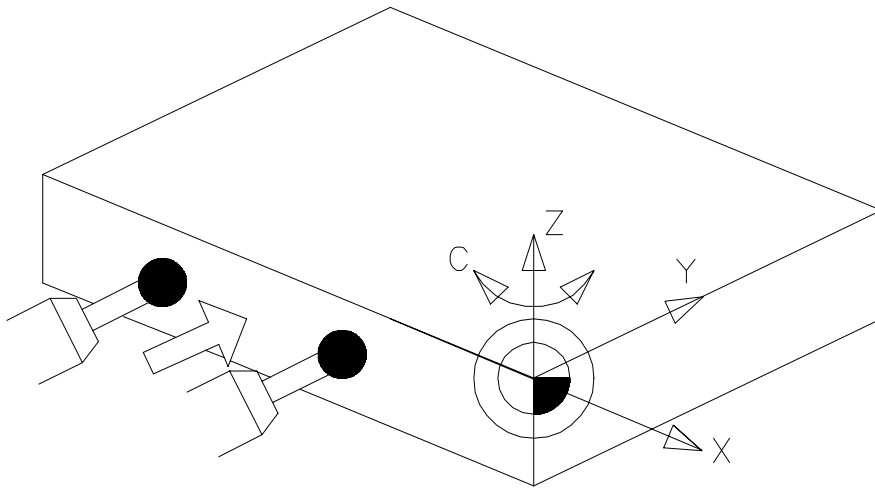
- C-Axis C1:** Aligning of a workpiece, mounted on a round table (C-axis) turning around the Z-axis, the measurement of two point in X direction are sufficient.
- The rotations angle is in respect with the X-axis.
 - The workpiece is turning around the Z-axis.
 - The measuring tracer stands in the Z-direction.



- C-Axis C2:** This is an extended possibility of C1:
- 1 The C-axis is turned 90 degrees and rotates around the Y-axis, instead of round the Z-axis.
 Alignment of a workpiece, mounted on a round table (C-axis) turning around the Y-axis, the measurement of two point in X direction are sufficient.
 - The rotations angle is in respect with the X-axis.
 - The workpiece is turning around the X-axis.
 - The measuring tracer stands in the Z-direction.



- 2 Alignment of a workpiece, mounted on a round table (C-axis) turning around the Z-axis, the measurement of two point in X direction are sufficient.
- The rotations angle is in respect with the X-axis.
 - The workpiece is turning around the X-axis.
 - The measuring tracer stands in the Y-direction.



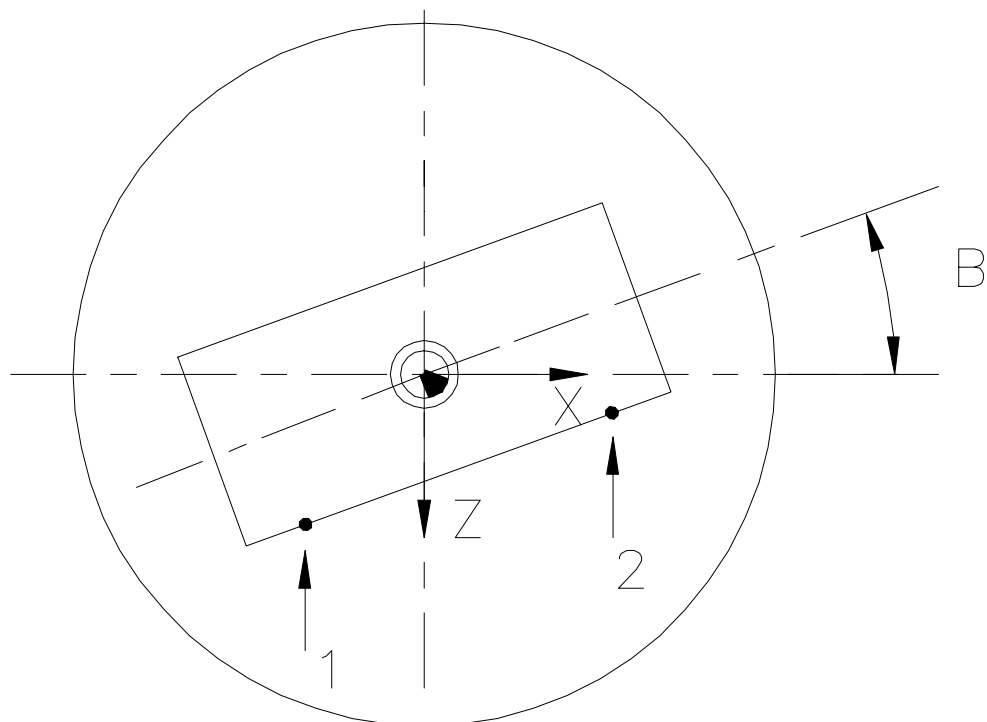
ALIGNING A WORKPIECE ON A ROTARY TABLE (B1=, C1=)

If a workpiece is mounted on a table, which rotates around the Y- or Z-axis, it is possible to align the workpiece by measuring two points in the X- and Z- or X- and Y-direction.

The angle the workpiece makes with the X-axis or Y-axis is automatically calculated by the control and can be used to rotate the table, so that the workpiece is parallel to the X-axis or Y-axis.

If the workpiece makes initially an angle with the X-axis, this angle can be programmed with the word B1= or C1=.

If B1= or C1= is not programmed, B1=0 or C1=0 is assumed.



TO CHANGE TOOL DIMENSIONS (T)

With the G50 function new tool dimensions derived from the measured corrections can be stored in the Tool Memory.

MULTIPLICATION FACTOR FOR TOOL DIMENSIONS (I, J, K, X1=)

The multiplication factor for the tool radius is X1=.

The multiplication factor for the length correction depends on the active main plane defined by G17, G18 or G19:

K... for Z-difference (G17-plane is active)

J... for Y-difference (G18-plane is active)

I. for X-difference (G19-plane is active)

The multiplication factor can have a positive or negative value.

If no factor is stated, the default value +1 will be used automatically by the CNC.

Examples**EXAMPLE 1. Changing a stored zero offset**

N... G50 X1 I0.8 N=54

Change the X-coordinate of the G54 offset by multiplying the correction by 0.8 and storing the new G54 X-coordinate value into the offset memory.

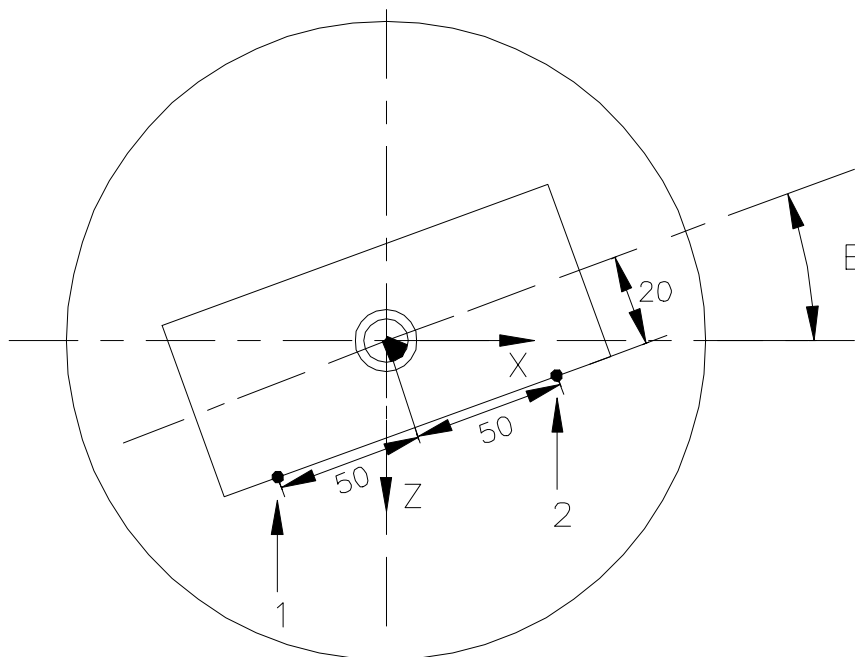
EXAMPLE 2. Changing a tool dimension

N... G50 T5 L1=1 K0.97 R1=1

Change the length of tool 5 by multiplying the Z-difference (tool in Z-axis) by 0.97, and store the new dimension into the tool memory.

EXAMPLE 3. Aligning a workpiece mounted on a rotary table

A part is mounted on a rotary table and should be aligned parallel to the X-axis. With a touch trigger probe two points on the part are measured and then the table is rotated over the calculated angle. The Controller knows, when G45 is activated twice and G50 once that a rotatory table has to be measured.



```

N50003
N1 G17
N2 G54
N3 T1 M6
N4 M27
N5 G45 X-50 Y-20 Z0 C0 J1
N6 G45 X50 Y-20 Z0 J1
N7 G50 C1 N=54
N8 M28
N9 G54

```

N10 G0 Z100 C0

Explanation

N1: Set the plane of operation

N2: Set the zero point

N3: Load the touch trigger probe

N4: Activate probe

N5: Measure point 1

N6: Measure point 2

N7: Update of the zero offset value of the C-axis with the calculated angle

N8: Deactivate probe

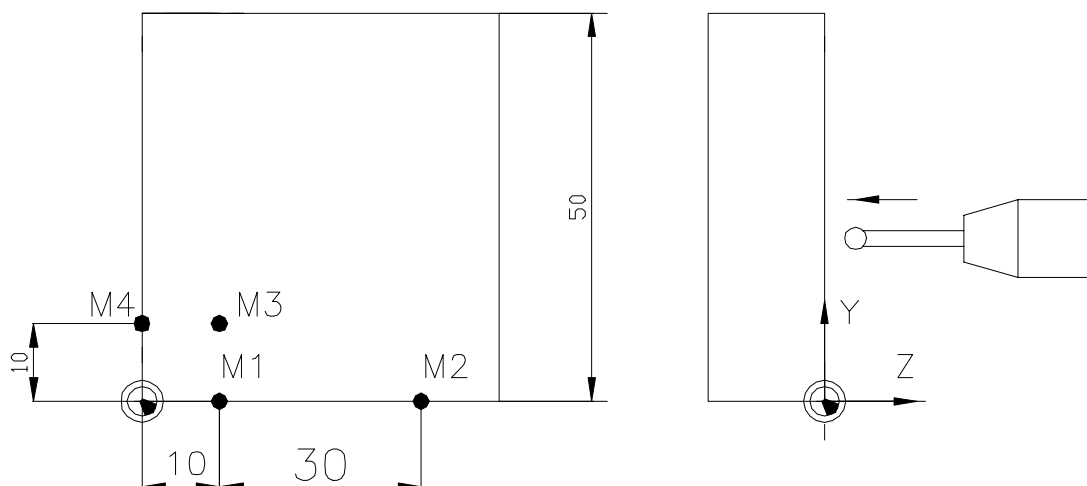
N9: Set the zero point

N10: Retract the tool and rotate the table to C0.

Note: N7 G50 C1 N=54 C1=30

If in block N7 C1=30, then the table rotates 30 Grad extra so that the table is parallel to the X-Axis.

EXAMPLE 4. Determining the zero point



The probe is standing in the Z-axis. The part is mounted on a table rotating around the Z-axis. Five points of the part (M1 to M4 and again M1) are measured. M1 and M2 for covering the angular displacement; M3, M4 and M1 for measuring the positions of the axes.

The section of the partprogram for determination the zero point could be:

N50004

N1 G54

N2 G17

N3 G0 X10 Y-10 Z10 T1 M6

N4 M27

N5 G45 X10 Y0 Z-5 C0 J1

```

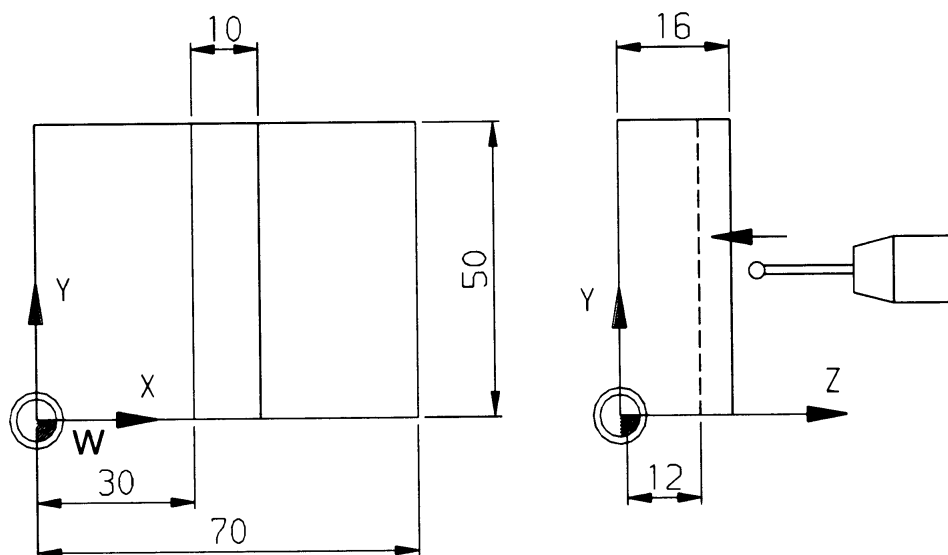
N6 G0 Z10
N7 G45 X40 Y0 Z-5 J1
N8 G0 Z10
N9 G50 C0 N=54
N10 G54
N11 G0 C0
N12 G45 X10 Y10 Z0 K-1
N13 G0 Z10
N14 G45 X0 Y10 Z-5 I1
N15 G0 Z10
N16 G45 X10 Y0 Z-5 J1
N17 G0 Z50
N18 G50 X1 Y1 Z1 N=54
N19 G54
N20 M28
:

```

Explanation

N1: Set the zero point
 N2: Set the plane of operation to be the XZ-plane
 N3: Load the touch trigger probe and move to the programmed position
 N4: Activate probe
 N5: Measure point M1
 N6: Retract the probe to avoid collision
 N7: Measure point M2
 N8: Retract the probe to avoid collision
 N9: Update of the zero offset value of the C-axis with the calculated angle
 N10: Set the zero point
 N11: Rotate the table to C0.
 N12: Measure point M3 to determine the position in the tool axis
 N13: Retract the probe to avoid collision
 N14: Measure point M4 to determine the position in the X-axis
 N15: Retract the probe to avoid collision
 N16: Measure point M1 to determine the position in the Z-axis
 N17: Retract the probe to avoid collision
 N18: Update of the zero offset values of the X-, Y- and Z-axis
 N19: Set the updated zero point
 N20: Deactivate probe

EXAMPLE 5. Correcting the length of a tool



With a mill a groove is made, the depth of the groove is measured and the tool length of the mill updated.

```

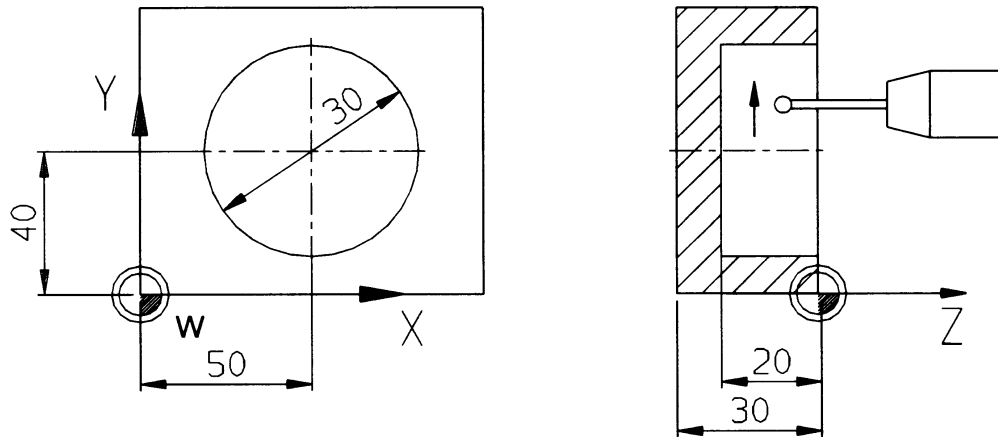
N90005
N1 G17
N2 T1 M6 (Mill radius 5 mm)
N3 X35 Y60 Z12 S1000 M3
N4 G1 Y-10 F200
N5 G0 Z200 M5
N6 T2 M6 (Probe)
N7 M27
N8 G45 X35 Y25 Z12 K-1
N9 G50 T1 L1=1
N10 M28
N11 Z200 M30

```

Explanation:

N1: Set the plane of operation to be the XY-plane
 N2: Load the mill of 10 mm diameter
 N3: Start the spindle and move the mill to the start point of the groove
 N4: Mill the groove
 N5: Retract the tool and stop the spindle
 N6: Load the probe
 N7: Activate probe
 N8: Measure the point in the negative direction of the tool axis
 N9: The calculated difference in the Z-axis is used to correct the length of tool 1
 N10: Deactivate probe
 N11: Retract the probe and end of program

EXAMPLE 6. Milling and measuring a hole



A hole is milled and measured with a touch trigger probe. Checks are provided to see if the tolerance on the radius of the hole is within the required limits. If the radius is too small, the hole is milled again. If the radius is too large the part is rejected and a message displayed.

The partprogram could be:

```

N50006
N1 G54
N2 G17
N3 T1 M6 (Mill radius 5 mm)
N4 G89 Z-20 B2 R15 K6 F300 S1000 M3
N5 G79 X50 Y40 Z0
N6 G0 Z50 M5
N7 T2 M6 (probe)
N8 M19
N9 M27
N10 G46 X50 Y40 Z-10 R15 I1 J1 F500 E5
N11 G0 Z50
N12 G49 R.02 R1=2 N=15 E5 (Hole > (15+.02) jump to N=19)
N13 G49 R2 R1=.02 N=18 E5 (Hole < (15-.02) jump to N=15)
N14 G29 E1 E1=1 N=21 (jump to the end of program)
N15 G50 T1 R1=1
N16 M28
N17 G14 N1=3 N2=6
N18 G29 E1 E1=1 N=21
N19 M0 (HOLE OUT OF TOLERANCE)
N20 M30

```

Explanation:

N1: Set the zero point
 N2: Set the plane of operation to be the XY-plane
 N3: Load a mill with a diameter of 10 mm
 N4: Define the fixed cycle for milling the hole
 N5: Mill the hole
 N6: Retract the tool and stop the spindle
 N7: Load the touch trigger probe
 N8: Spindle stop at a certain angle
 N9: Activate probe

- N10: Measure the hole at four points
- N11: Retract the probe to avoid collision
- N12: Check to see if the radius of the hole is not too big, (less than $15+.02$). If the radius is too big, reject the part and display a message to it.
- N13: Check to see if the radius of the hole is not too small, (greater than $15-.02$). If the radius is too small, update the radius value in the tool memory and mill the hole again.
- N14: Jump to the end of the program
- N15: Update of tool radius in tool memory
- N16: Deactivated probe
- N17: Repeat of the blocks N3 to N6 to mill the hole within tolerance.
- N18: Jump to the end of the program
- N19: Stop the program execution and display a message
- N20: End of program

5.28 G51/G52 Cancel/activate pallet zero point shift

Purpose

Fix pallet zero offset with programmed value.

Format

Activate and deactivate after G51 has been used:

N... G52

To cancel:

N... G51

Parameter

none

Notes and usage

The function G52 can be cancelled with softkey CLEAR CONTROL or overridden by programming G51 .

The functions G51 and G52 will stay active after Softkey CANCEL PROGRAM, M30 or turning off the controller.

If there is a zero point shift G54.[nr] active. Then G52 is active with the zero offset. Is G52 active and G54.[nr] are activated then G54.[nr] are activated with de zero offset of G52.

Function G52 is used for the purpose of automation, for instance pallet control. In this case the values for G52 are set by an IPLC program.

When **MC84 = 0** then G52 is stored in ZO.ZO (zero point).

When **MC84 > 0** then G52 is stored in PO.PO (Pallet Offset).

In ZO.ZO and in PO.PO the zero points can be edited.

5.29 G53/G54—G59 Cancel/activate zero point shift

On two ways the zero point shift can be done:

- 1) MC84=0 with G53/G54—G59
- 2) MC84>0 with G53 and G54.<Nr.>

Apart from the present zero point shift table G54..G59 there is another zero point shift table G54 I[nr] showing a maximum of 100 zero point shifts. The appropriate zero point shift is selected by machine constant MC84.

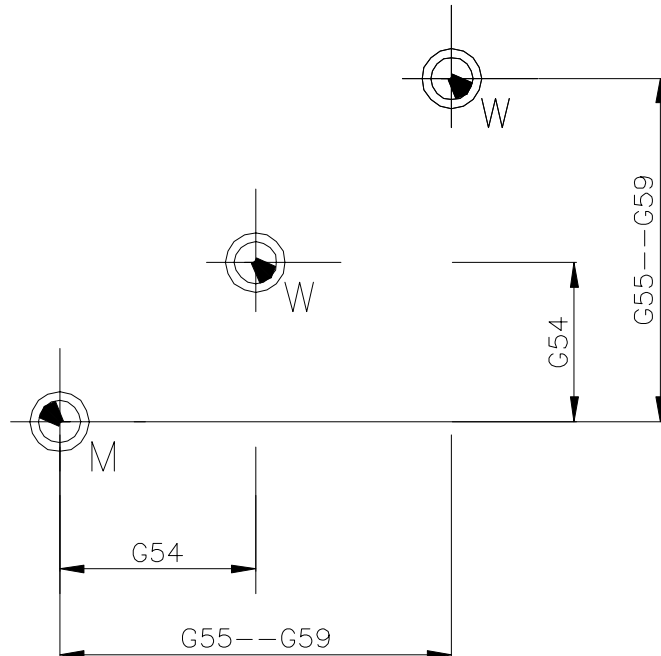
The functionality is the same as that of the present zero point shift memory G54..G59, except for the following extensions and differences:

- 100 potential zero point shifts in the zero point shift memory
- Identification of zero point memory Ze.Ze (MC84 > 0)
- Programming (shift values) of zero point shift in NC program
- Programming of angle of rotation (B4=) in zero point shift
- Programming of zero point shift with an index (G54 I[nr])
- Comment is entered in zero point shift memory

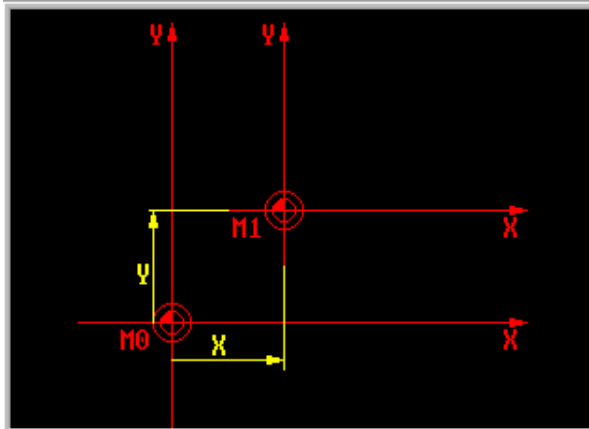
5.29.1 G53/G54—G59 Cancel/activate zero point shift (MC84=0)

Purpose

To shift the workpiece zero point to a new position whose coordinate values are stored in the zero point memory (under the appropriate number).



- G53 Cancels the active G54-G59 zero point
 G54 Activates G54 zero point shift
 :
 G59 Activates G54 zero point shift

**Format**

To activate:

N... [G54/G55/G56/G57/G58/G59]

To cancel:

N... G53

Parameters

None

Modal words

E...=, T2=

Type of function

Modal

Associated functions

G51/G52, G92, G93

Notes and Usage**MACHINE ZERO POINTS**

If a machine has several clamping stations or more than one rotary table it is necessary to state secondary machine zero points. These points are related to the geometric machine zero point (M_0). The axial distances measured from M_0 specify the position of these secondary zero points and are stored in the Zero Offset Memory together with their identifying G-function.

THE ZERO OFFSET MEMORY

All values for the G54-G59 zero point shifts (zero offset values) must be stored in the appropriate memory by using the user's panel or a data carrier before the program is executed.

G52 ZERO POINT SHIFT

The function G52 is not influenced by one of the functions G53 to G59.

ABS./INCR. ZERO POINT SHIFTS G92/G93

A programmed zero point shift (G92 or G93) is cancelled by one of the functions G53 to G59.

SCALING, MIRROR IMAGE AND AXIS ROTATION (G73, G92/G93)

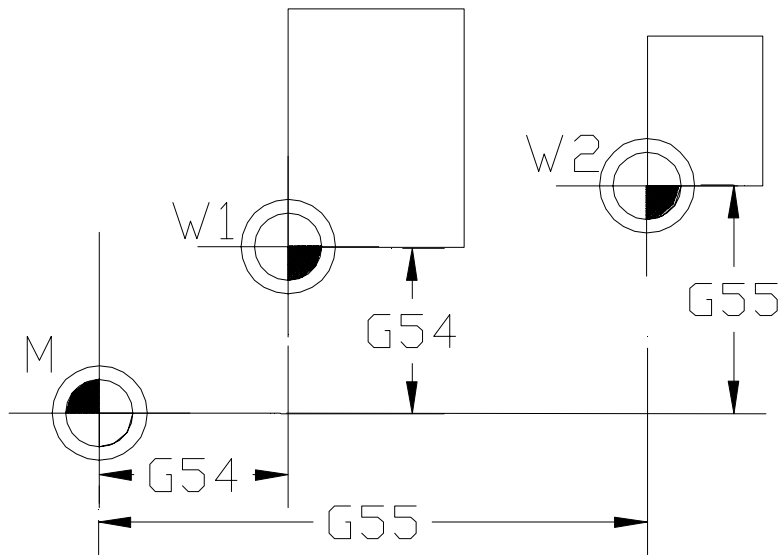
One of the functions G53 to G59 can be used in a program sequence, which is scaled, mirrored or rotated. The zero point shift is performed in the coordinate system of the machine tool and not influenced by the programmed change of coordinates.

CANCELLATION

A secondary machine zero point can be overridden by programming G53.

G53 is automatically set at switching on the control and after a reference point search.

The functions G54 to G59 are not cancelled by CLEAR CONTROL, M30 or Softkey CANCEL PROGRAM.

Example

```

N60   G54
:
N600  G55
:
N700  G53

```

Explanation:

N60: Select the secondary machine zero point W1. Its coordinates (X40, Y100, Z300) are retrieved from the Zero Offset Memory.

All programmed coordinates are measured from W1.

N600: Select the secondary machine zero point W2. Its coordinates (X200, Y100, Z100) are retrieved from the Zero Offset Memory.

Machine zero point W1 is cancelled and W2 is now active, therefore all programmed coordinates are measured from W2.

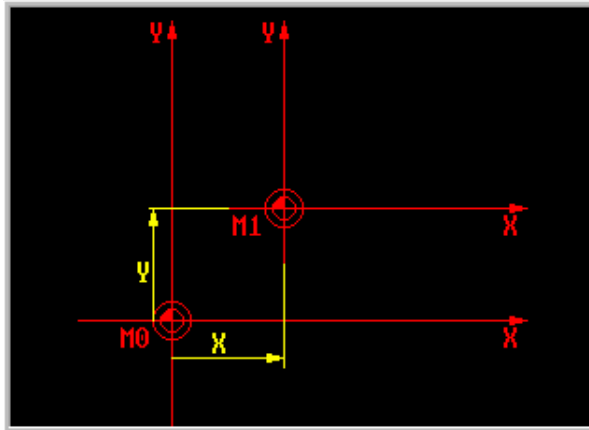
N700: Cancel with G53 machine zero point W2. Then the coordinates (X0, Y0, Z0) are cleaned.

Zero point W2 will be deleted and M (machine zero point) will be active. After this action all coordinates are measured from M.

5.29.2 G54 Extended zero point shift (MC84>0)

Purpose

To shift the workpiece zero point to a new position. The coordinate values can be entered in the zero point shift memory or programmed in the NC program block.



Format

Define and use zero point shift as follow:
G54 I[nr] [Axis coordinates] {B4=..}

Use zero point shift as follow:
G54 I[nr]

To cancel
N... G53

Parameter

```
G    Activate zero point shift
X    Zero point coordinate
Y    Zero point coordinate
Z    Zero point coordinate
A    Zero point angle
B    Zero point angle
C    Zero point angle
I    Zero point index
B4=  Angle of rotation absolute
```

A zero point shift can contain up to 6 Axis coordinates.

Type of function

Modal

Associated functions

G50, G51, G52, G53, G54 ... G59, G92, G93, G149, G150

Notes and usage**NUMBER OF ZERO POINTS**

The number of possible zero point shifts in the table is determined by a machine constant (MC84). (0 < MC84 < 99).

CHANGING MACHINE CONSTANT MC84

The zero point shift table is adjusted in the event of scaling (MC84 > 0). The existing zero points is maintained. Extended zero points are initialised to zero.

Attention:

If MC84 is zeroed, the table is changed (ZE.ZE is changed to ZO.ZO). The new zero point table is initialised to zero.

ENTERING IN ZERO POINT SHIFT MEMORY

Shift values can be entered in the zero point memory in two different ways:

- The values of zero point shifts G54 I[nr] are entered into the zero point shift memory via the control panel or through a data carrier before the program is executed.
- The values of zero point shift G54 I[nr] X.. Y.. Z.. A.. B.. C.. B4=.. are programmed in an NC program block. When the program is edited, the programmed values are accepted in the zero point shift memory and activated.

Attention:

If no new zero point shift values have been programmed in the program block, the zero point shift values already stored in the memory are not overwritten or deleted. The axis coordinates not programmed are taken from the memory. Risk of collision!

COMMENT

Additionally, each zero point shift in the table may be commented.

AXIS ROTATION

Additionally, each zero point shift in the table may involve an axis rotation.

First, the shift is executed and then the coordinate system is rotated through angle B4=.

MACHINE ZERO POINTS

If a machine has several clamping stations or more than one rotary table it is necessary to state secondary machine zero points. These points are related to the geometric machine zero point (M0). The axial distances measured from M0 specify the position of these secondary zero points and are stored in the Zero Offset Memory together with their identifying G-function.

ZERO POINT SHIFT MEMORY

All values of zero point shifts G54 I[nr] should be stored in the zero point shift memory via the control panel or by a data carrier, before the program is executed.

G52 ZERO POINT SHIFT

G52 does not affect the functions G53...G59. If G52 is active, G54..G59 will be effective from this shift onwards.

ABSOLUTE/INCREMENTAL ZERO POINT SHIFTS (G92/G93)

A programmed zero point shift (G92 or G93) is deleted from any of the G54 I[nr] functions.

SCALING, MIRRORING AND AXIS ROTATION (G73, G92/G93)

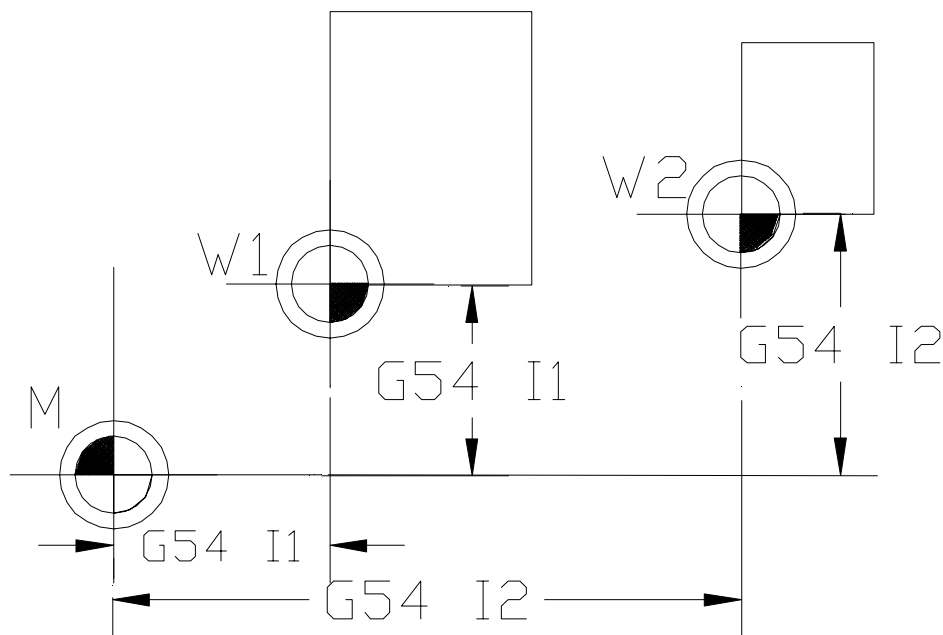
It is allowed to use any of the functions G54 I[nr] in a program section which is to be scaled, mirrored or rotated. The zero point shift takes place in the coordinate system of the machine tool and is not affected by the programmed change of coordinates.

DELETE

G54 I[nr] is automatically deleted by the CLEAR CONTROL Softkey and by programming G53.

The functions G54...G59 are not deleted when the CANCEL PROGRAM Softkey or M30 is used.

Example



```

N60 G54 I1
:
N600 G54 I2
:
N700 G53

```

Explanation:

N60: Select the secondary machine zero point W1. Its coordinates (X40, Y100, Z300) are retrieved from the Zero Offset Memory.

All programmed coordinates are measured from W1.

N600: Select the secondary machine zero point W2. Its coordinates (X200, Y100, Z100) are retrieved from the Zero Offset Memory.

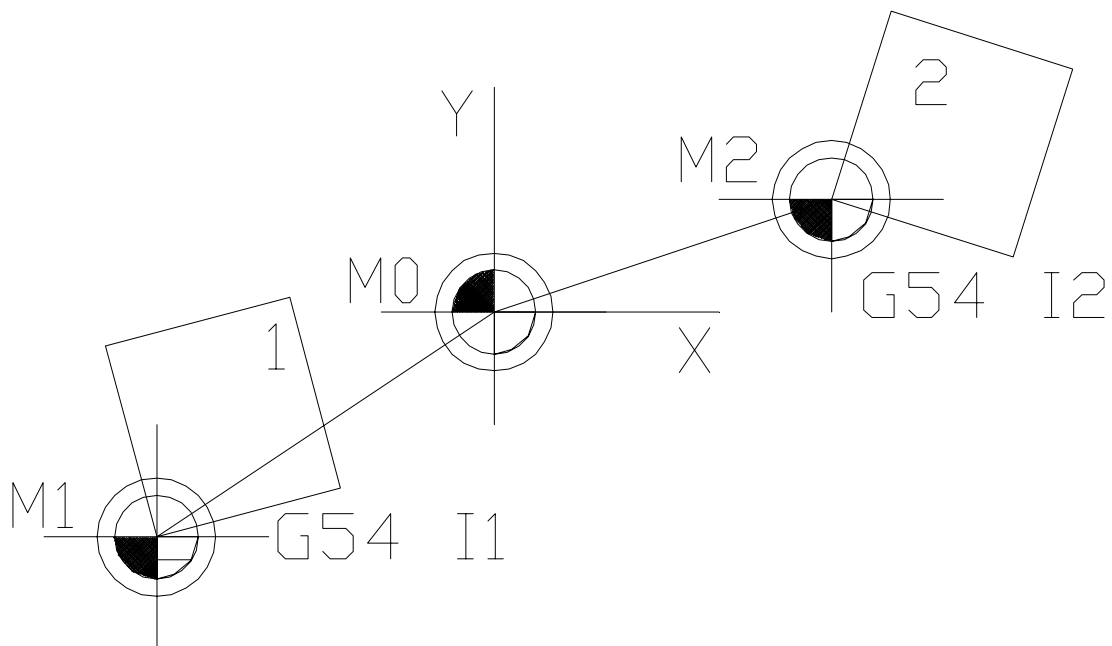
Machine zero point W1 is cancelled and W2 is now active, therefore all programmed coordinates are measured from W2.

N700: Cancel with G53 machine zero point W2. Then the coordinates (X0, Y0, Z0) are cleaned.

Zero point W2 will be deleted and M (machine zero point) will be active. After this action all coordinates are measured from M.

Example

Axis rotation



1= Workpiece 1
2= Workpiece 2
3= Table

Entry in zero point table and calling:

N60 G54 I1 X-42 Y-15 B4=14 (Z0 C0)

Zero point shift values are entered in the zero point shift table.

Machine workpiece 1, all programmed coordinates are measured from M1.

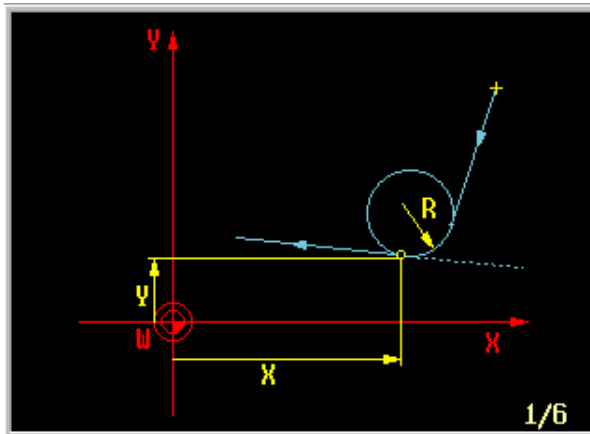
N120 G54 I2 X10 Y24 B4=-17

Machine workpiece 2, all programmed coordinates are measured from M2.

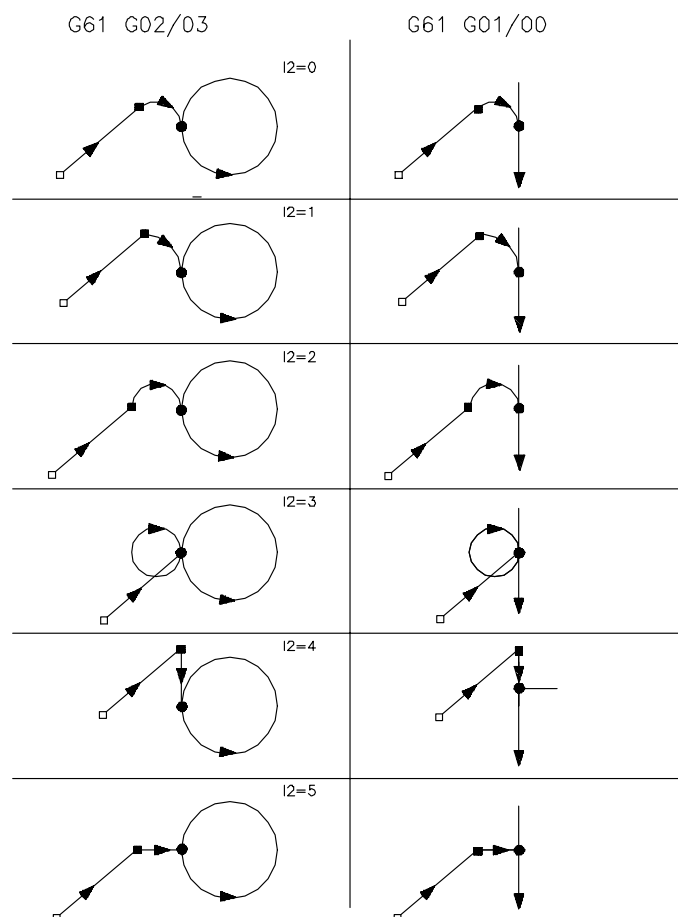
5.30 G61 Tangential approach

Purpose

Programs a tangential approach movement between a starting point and start of a contour.



Format



TANGENTIAL APPROACH MOVEMENT TO THE CONTOUR G61

□ Actual position.

■ Calculated start point in main plane. Z1 can be programmed. If Z1 is not programmed then Z1=Z.

● Start point contour (X, Y, Z).

N... G61 {I2=} X... Y... Z... R... [{Z1= | Y1= | X1=}] {I1=} {F2=}

N... G61 {I2=} B2=... L2=... Z... R... [{Z1= | Y1= | X1=}] {I1=} {F2=}

Parameters

```
G    Tangential approach
X    Endpoint tangential approach
Y    Endpoint tangential approach
Z    Endpoint tangential approach
P    Point definition number
R    Radius
Z1=  Startpoint in Z
B2=  Polar angle
?90= Endpoint abs. (X,Y,Z..)
?91= Endpoint incr. (X,Y,Z..)
I1=  Linear movement 0=rapid,1=feed
I2=  Tangential approach definition
L2=  Polar length
```

Z1= Start height calculated begin point in Z
 I1= Linear movement 0=rapid, 1=feed.
 I2= Approach method
 I2=0 with line and tangent circle
 I2=1 with quarter circle
 I2=2 with semicircle
 I2=3 Helix for feeding (for pockets)
 I2=4 Parallel to contour
 I2=5 Vertical

Modal words

F, S, T, T1=, T2=, M, H, E.=
 S1=, M1=

Type of function

Not modal

Associated functions

G0, G1, G2, G3

Notes and usage

STARTPOINT OF THE APPROACH MOVEMENT

The Controller calculates the start point. The first movement is a positioning movement towards the calculated start point. From this position the approach movement towards the contour is started.

HIGHT OF THE STARTPOINT IN TOOL-AXIS

The Z (G17) parameter contents the contour start point in the tool-axis. The parameter Z1 (G17) contents the height of the start point of the approach movement. By a difference of Z and Z1 a slanting movement is made.
(In G18 Y and Y1=)

APPROACH MOVEMENT

The approach movement consists of two movements. The first movement is a rapid or a feed movement (depends if is I1=0 or I1=1) towards the calculated start point. The second movement is a feed movement to the starting point of the contour.

APPROACH SIDE

The parameters G41 or G42 determine the approach side. (G41 is left, G42 is Right). When G40 is active the approach is the same as the in G41 (left side).

THE CIRCULAR ARC MOVEMENT

The circular movement is determined by the position of the starting point of the contour and the starting point of the tangential movement. The Direction of the tangential circular movement is determined by the direction of the contour movement. The movement is always fluid between the tangential circular arc and the contour movement.

RADIUS COMPENSATION

The radius compensation (G41/G42) must be activated right before G61 is active the compensation will be activated during the linear movement. The actual position determinates the calculated position.

If the radius compensation is active the linear movement and the circular movement are operated according to the radius compensation.

PERPENDICULAR APPROACH MOVEMENT

The position of the contour start point determines the position of the perpendicular approach movement.

G1-FUNCTION

If there is no G-function programmed after a G17 Block G1 will not be active.

LIMITATION WHEN I2=0 IS ACTIVE

When the actual position is further away then one diameter from the circular movement then the approach movement contents a linear and a circular movement. When the actual position is within the circular movement then parameter I2= changes from 0 to 1, and the begin movement contents a quarter circle movement.

LIMITATIONS

Programming a G61 has the following limitations:

- G61 is in G64 mode NOT allowed
- G61 is in MDI NOT allowed
- G61 is in G182-Mode NOT allowed

The program block after G61 has limitation. The following functions are allowed:

- G64
- G0, G1, G2, G3 with movements in de active plane

Remark:

From V410 the result of the G61, comparing with earlier versions, will be different.

Notes:

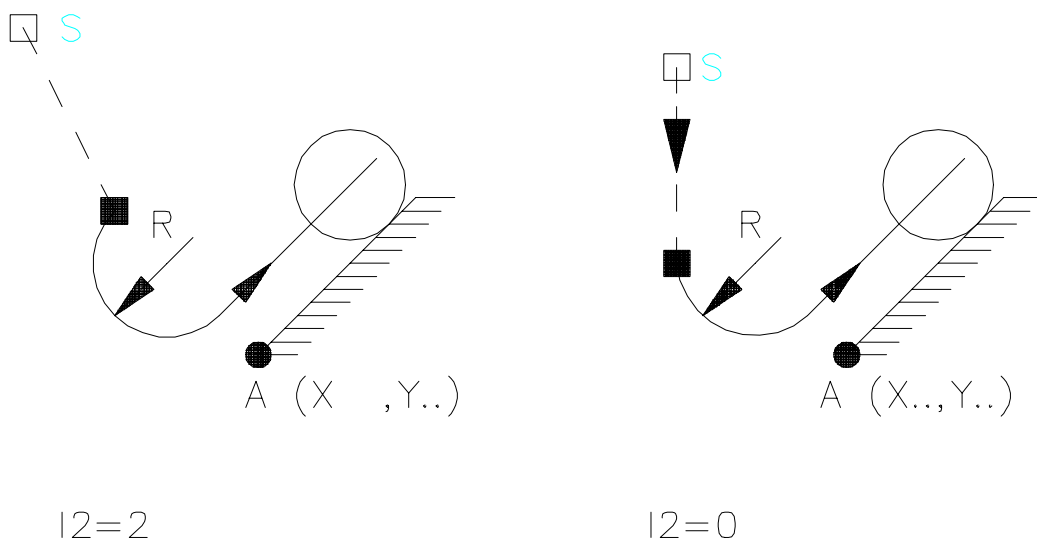
The programmer must program the approach movement in such away that during milling the contour doesn't damage.

SUPPORT PROGRAMMING

The functions "tangential approach" (G61) and "tangential exit" (G62) can be used in the following mode:

- FREE ENTRY
- SUPPORT ENTRY

The SUPPORT ENTRY supports the programmer with pictures and text.

Example:

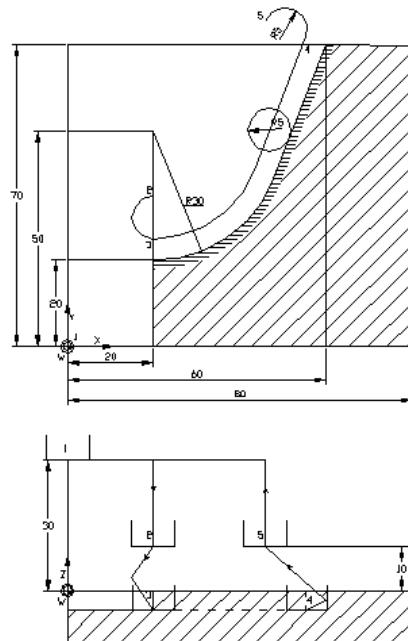
```

N33 G17
N34 G0 X... Y... (S)
N35 G41
N36 G61 I2=. X... Y... I1=0 R5
N37 G64

```

Explanation:

- N33: Define the main plane. All the movement occur in one plane.
- N34: Moving to the start point position.
- N35: Radius compensation must be active for the left side.
- N36: Tangential approach movement. This movement contents two movements. First a linear movement to the calculated position (with rapid (I1=0) or feed (I1=1)) and then a circular movement.
- N37: Start a contour.

Example:

```

N1 G17
N2 T1 M6 (Mill R5)
N3 F500 S1000 M3
N4 G0 X0 Y0 Z50
N5 G41
N6 G61 I2=2 X20 Y20 Z-5 Z1=10 R5 I1=0 F2=200
N7 G64
N8 G3 I20 J50 R1=0
N9 G1 X60 Y60
N10 G63

```

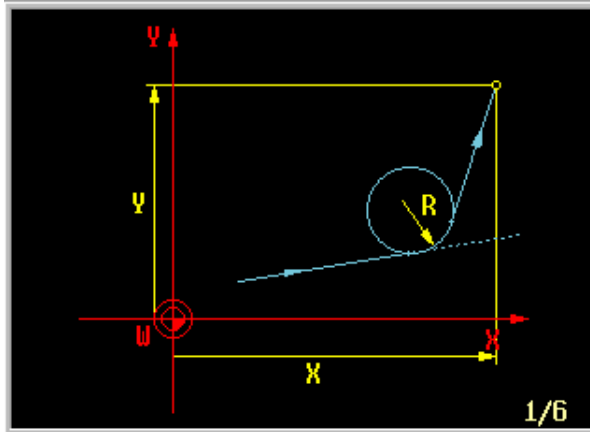
Explanation:

- N1: Active XY-plane (G17).
 N2: Load Tool T1.
 N3: Activating feed, speed and spindle rotation (M3).
 N4: Move tool rapidly to programmed position (position 1: X0 Y0 Z30).
 N5: Set radius compensation LEFT
 N6: -Tangential approach movement
 -I2=2 is a semicircle
 -The first section is a rapid movement with positioning logic to the start point of the semicircle movement (position 2: X.. Y.. Z10). The Radius compensation is activated in this movement.
 -The circular movement will perform a helix movement.
 The contour starts with position X20 Y20 Z-5. (Position 3: X20 Y25 Z-5).
 N7: Start the contour description.
 N8: A circular movement tangential to a Line.
 N9: Tangential Linear movement.
 N10: End of the contour description.

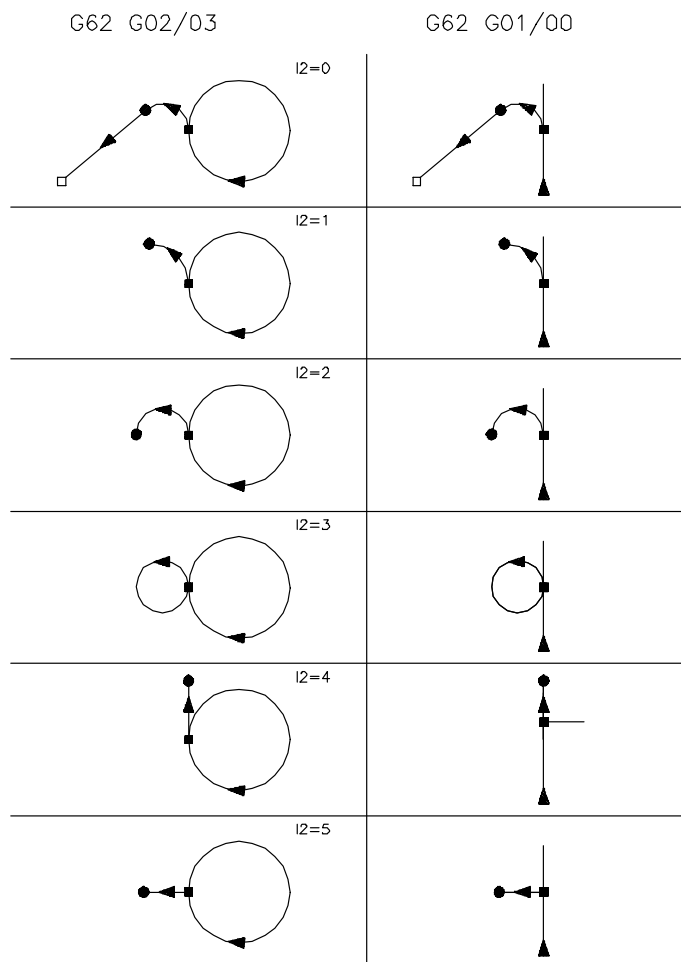
5.31 G62 Tangential exit

Purpose

Programs a tangential exit after the end of the contour.



Format



TANGENTIAL EXIT FROM DE CONTOUR G62:

- ☐ End point contour.
- ☒ Calculated end point in main plane. Z1 can be programmed.
(If Z1 is not programmed then the value doesn't change.)
- ☒ Programmed end point tangential exit (X, Y, Z) (only I2=0).

N... G62 I2>0 Z1=... R... {I1=} {F2=}
 N... G62 I2=0 X... Y... Z... Z1=... R... {I1=} {F2=}
 N... G62 I2=0 B2=... L2=... Z... R... {I1=} {F2=}

Parameter

G Tangential exit
 X Endpoint tangential exit
 Y Endpoint tangential exit
 Z Endpoint tangential exit
 P Point definition number
 R Radius
 Z1= Startpoint in Z
 B2= Polar angle
 ?90= Endpoint abs. (X,Y,Z..)
 ?91= Endpoint incr. (X,Y,Z..)
 I1= Linear movement 0=rapid,1=feed
 I2= Tangential exit definition
 L2= Polar length

Z1=

I1=

I2=

I2=0 with circle with tangent line.

I2=1 with quarter circle

I2=2 with semicircle

I2=3 Helix for feeding (for pockets)

I2=4 Parallel to contour

I2=5 Vertical

Modale Parameter

F, S, T, T1=, T2=, M, H, E..=

S1=, M1=

Type of Function

Not modal

Associated functions

G0, G1, G2, G3

Note:

To understand the function G62 read first function G61.

Notes and usage**CANCEL RADIUS COMPENSATION**

The radius compensation is deactivated in the G62 line. The movement to the calculated position will be done with radius compensation.

Remark:

From V410 the radius compensation will be deactivating in G62.

Remark:

From V410 the result of the G62, comparing with earlier versions, will be different.

LIMITATIONS

Programming a G62 has the following limitations:

- G62 is in G64 mode NOT allowed
- G62 is in MDI NOT allowed
- G62 is in G182-Mode NOT allowed

The program block after G61 has limitation. The following functions are allowed:

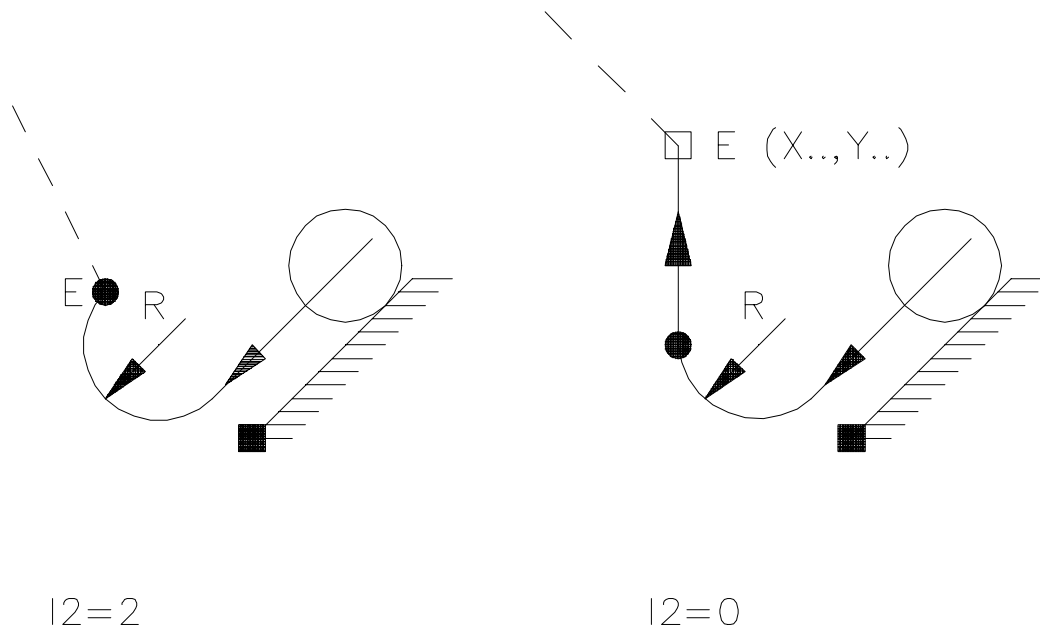
- G64
- G0, G1, G2, G3 with movements in de active plane

SUPPORT PROGRAMMING

The functions "tangential approach" (G61) and "tangential exit" (G62) can be used in the following mode:

- FREE ENTRY
- SUPPORT ENTRY

The SUPPORT ENTRY supports the programmer with pictures and text.

Example

```

N51 X.. Y..
N52 G63
N53 G62 I2=2 R5 or G62 I2=0 X.. Y.. R5

```

Explanation:

N51: Last movement in the contour description.

N52: End contour description.

N53: (I2=2) Tangential Exit. This movement contains a circular movement.

(I2=0) Tangential Exit. This movement contains two movements a circular and a linear movement.

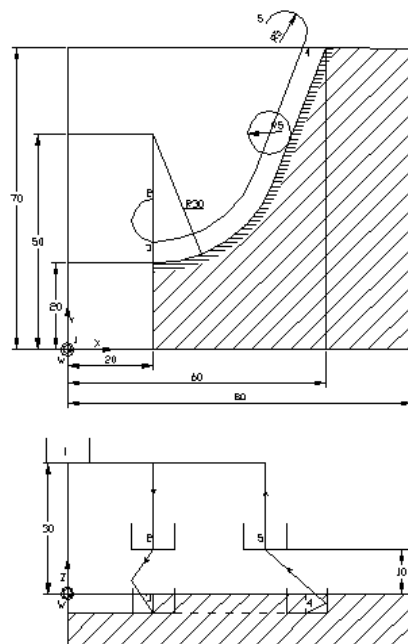
The radius compensation is deactivated between the circular and the linear movement.

Example

```

N1 G17
N2 T1 M6 (Mill R5)
N3 F500 S1000 M3
N4 G0 X0 Y0 Z50
N5 G41
N6 G61 I2=2 X20 Y20 Z-5 Z1=10 R5 I1=0 F2=200
N7 G64
N8 G3 I20 J50 R1=0
N9 G1 X60 Y60
N10 G63
N11 G62 I2=2 Z1=20 R5
N12 G0 X0 Y0 Z50
N13 M30

```



Explanation:

- N1: Active XY-plane (G17).
 N2: Load Tool T1.
 N3: Activating feed, speed and spindle rotation (M3).
 N4: Move tool rapidly to programmed position (position 1: X0 Y0 Z30).
 N5: Set radius compensation LEFT
 N6: -Tangential approach movement
 -I2=2 is a semicircle
 -The first section is a rapid movement with positioning logic to the start point of the semicircle movement (position 2: X.. Y.. Z10). The Radius compensation is activated in this movement.
 -The circular movement will perform a helix movement.
 The contour starts with position X20 Y20 Z-5. (Position 3: X20 Y25 Z-5).
 N7: Start the contour description.
 N8: A circular movement tangential to a Line.
 N9: Tangential Linear movement with endpoint (position 4: x.. Y.. Z-5).
 N10: End of the contour description.
 N11: -Tangential Exit
 -I2=2 is a semicircle
 -The circular movement will perform a helix movement. Start point in the Z-axis is -5, and the endpoint is 10.
 Radius compensation will be deactivated.
 N12: Retract Tool (position 1: X0 Y0 Z30)
 N13: Program End.

5.32 G63/G64 Cancel/Activate geometric calculations

Purpose

- G63: To cancel the geometric calculations and to return to programming complete blocks.
 G64: To activate the geometric calculations.

General principles for using the geometry

Between the functions G64 and G63 a contour can be described. An easy way of programming linear and circular movements makes it possible to let the control perform the necessary calculations for e.g. an intersection point or point of tangency. Each time a calculation is required at least two blocks of data are used. Each block is programmed with the standard G-functions for linear (G0 and G1) and circular movements (G2 and G3) and some information to define the line or circle. These blocks do not necessarily contain all data as previously specified, but with some special words (indicators) is achieved, that the missing data can be calculated by the control.

The first block establishes where the start point is located and what type of end point is required. The second block supplies the data for calculating the end point coordinates of the first block as e.g. a point of tangency or an intersection point of two elements. This end point is also the start point of the second block.

Between these movements can be inserted:

- A chamfer (between linear movements),
- A rounding (between intersecting elements),
- A connecting circle (between tangent elements or elements which do not meet)

It may happen that the second block does not supply enough data for calculating the end point of the first block. In that case the control looks for the next block and try to calculate the end point of the second block and first one. Up to five blocks are looking for in advance.

Formats

Activating geometric calculations

N.. G64

Cancelling geometric calculations

N.. G63

Linear (G0/G1) and circular movements (G2/G3)

Only the most commonly used formats are given here. Refer to a special appendix at the end of this manual for a detailed description of the possible formats for G0/G1 and G2/G3 in the many cases and also for examples of the use of the geometry.

For all formats the G64-function is assumed to have already been programmed in a previous block and is therefore active.

The XY-plane is also assumed to be the active plane. Refer to Notes and usage PLANE SELECTION for changes to be made in the formats, if another plane is active.

Possible parameters between G64 and G63 lines.**Straight line**

```

G    Linear interpolation
X    Endpoint coordinate
Y    Endpoint coordinate
Z    Endpoint coordinate
I    Chamfer length
X1=  Arbitrary endpoint coordinate
Y1=  Arbitrary endpoint coordinate
B1=  Angle
B2=  Polar angle
I1=  Parallel shift
J1=  1=intersection left, 2=right
L2=  Polar length
P1=  Point definition number
R1=  R1=0 tangent to line

```

Circle

```

G    Circular clockwise
X    Endpoint coordinate
Y    Endpoint coordinate
Z    Endpoint coordinate
I    Center point in X / pitch in X
J    Center point in Y / pitch in Y
K    Center point in Z / pitch in Z
R    Circle radius
B1=  Angle
B2=  Polar angle
B3=  Polar angle for center
B5=  Angle of arc
J1=  1=intersection left, 2=right
L2=  Polar length

```

```

L3=  Polar length for center
P1=  Point definition number
R1=  R1=0 tangent to line

```

Modal words

F, F1=, S, T, H, E and M except M6, M66 and M67

Type of function

Modal

Notes and Usage

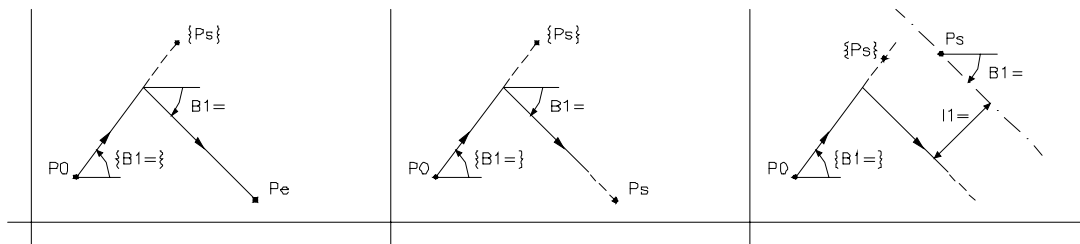
Within function G64 it is not possible to active G4.

Explanation of the possible formats:

In the illustrations in which the formats are explained, the following conventions are used:

P_0 = a start point known from the previous block
 P_s = a support point on a line or on a parallel line
 P_e = a programmed end point
M = a programmed circle centre point
R = a programmed radius of a circle

5.32.1 Intersection point between two straight lines



Intersection point with known start point from first line

Possible definition of the first line

N.. G1 {B1=..}

(Start point and angle)

N.. G1 X{Ps} Y{Ps}

(Start point and support point)

Possible definition of the second line

N.. G1 B1=.. X{Pe} Y{Pe}

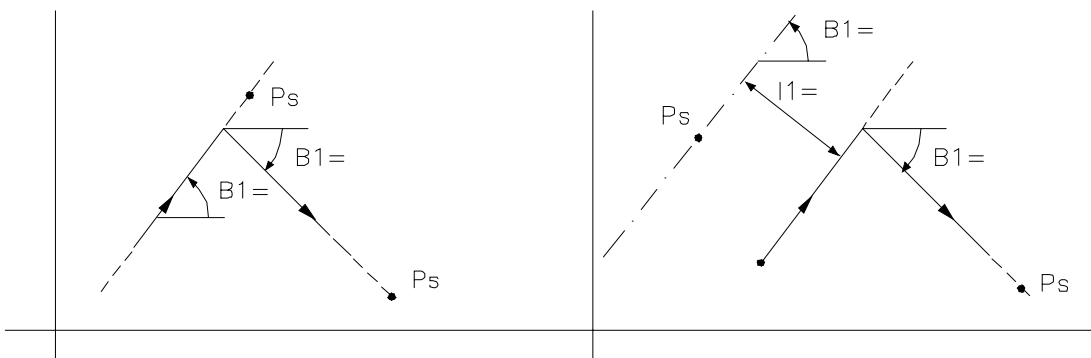
(Angle and end point)

N.. G1 B1=.. X{Ps} Y{Ps}

(Angle and support point)

N.. G1 X{Ps} Y{Ps} B1= l1=

(Support point, angle and parallel line)



Intersection point with unknown start point from first line

Possible definition of the first line

N.. G1 {B1=..} X{Ps} Y{Ps}

(Support point and angle)

N.. G1 B1= X{Ps} Y{Ps} l1=

(Support point, angle and parallel line)

Possible definition of the second line

N.. G1 B1=.. X{Pe} Y{Pe}

(Angle and end point)

N.. G1 B1=.. X{Ps} Y{Ps}

(Angle and support point)

N.. G1 X{Ps} Y{Ps} B1= l1=

(Support point, angle and parallel line)

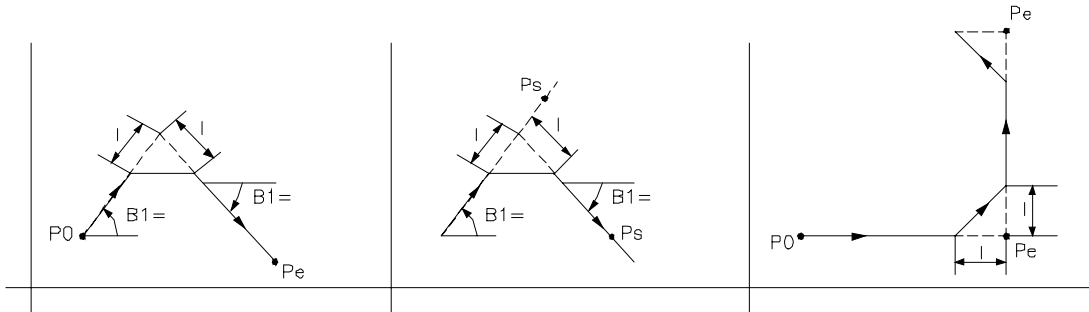
Note: It is also possible to program the intersection point as an end point, if it is known from the drawing. Refer to END POINT in Notes and usage for details.

A chamfer inserted between two intersecting lines

The lines are programmed as indicated in the previous section.

The chamfer is:

- symmetrically located around the intersection point



- programmed with G1 and the length of the chamfer (I-word)

Chamfer between two straight lines

N.. G1 {B1=..} or {support point/parallel line}

N.. I.. {chamfer parameter}

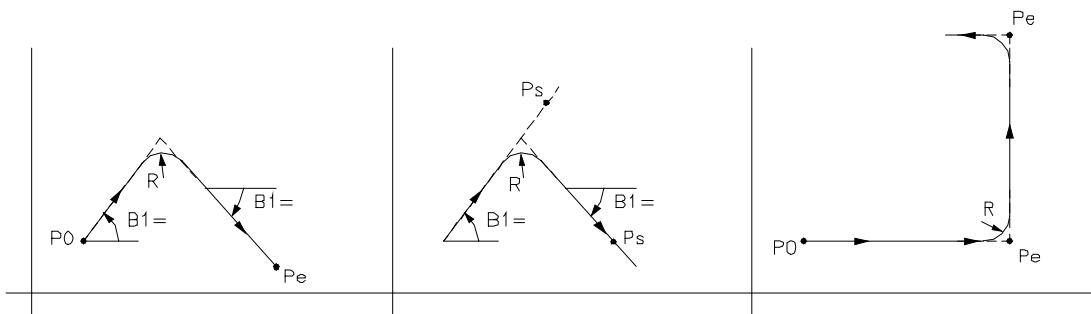
N.. B1=.. {end point} or {support point/parallel line}

A rounding between two intersecting lines

The lines are programmed as indicated in the previous section.

The rounding is:

- tangent to the line from the previous block and to the line of the next one.
- programmed with:
- G2 or G3 indicating the direction of movement,
- the radius (R-word) of the rounding.



Rounding between two straight lines

N.. G1 {B1=..} or {support point/parallel line}

N.. G2/G3 R..

N.. G1 B1=.. {end point} or {support point/parallel line}

Note: It is also possible to insert a rounding between a straight line and a chamfer or between a chamfer and a straight line.

N.. G1 {B1=..} or {support point/parallel line}
 N.. G2/G3 R..
 N.. I.. {chamfer parameter}
 N.. G2/G3 R..
 N.. G1 B1=.. {end point} or {support point/parallel line}

Intersection point indicator

If an intersection point between line and circle or two circles should be calculated, two points are possible.

With the word J1= is indicated which point is required:

- J1=1: the left intersection point (P1)
- J1=2: the right intersection point (P2).

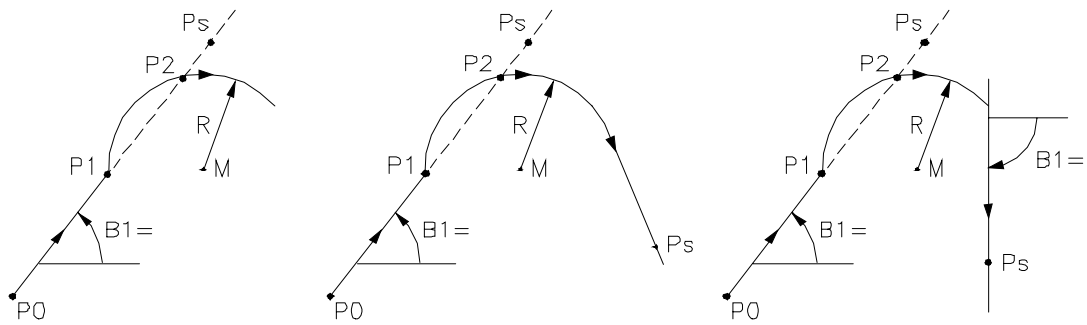
With a line through the circle centre point

- J1=1: the smallest distance between the start point or end point
- J1=2: the largest distance between the start point or end point.

With a line that starts or ends in the circle centre.

- J1=1: is the first intersection.
- J1=2: is the second intersection.

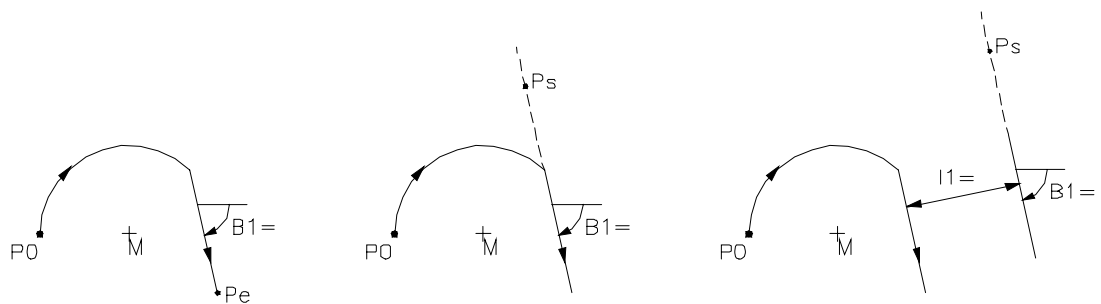
5.32.2 Intersection point between line and circle or circle and line



Line to circle

N..G1{B1=..} or {support point} J1=1/2

N..G2/G3 I..J.. R..

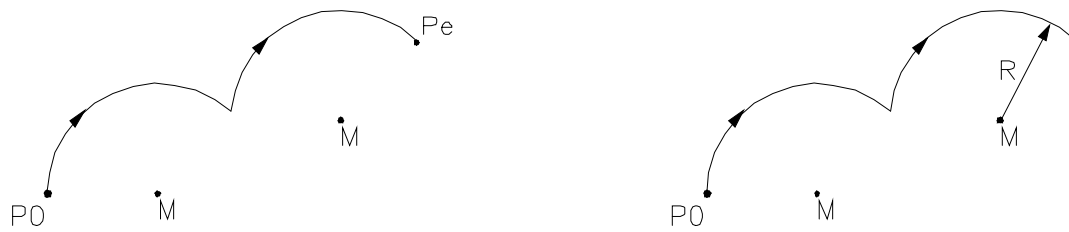


Circle to line

N.. G2/G3 I.. J.. R.. J1=1/2

N..G1 B1=..{support point} or {end point}

5.32.3 Intersection point between two circles



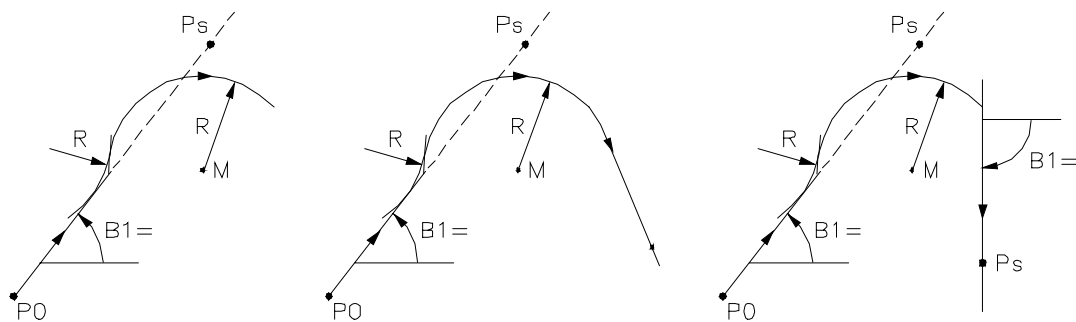
N..	G2/G3	I..	J..	{R..}J1=1/2
N..	G2/G3	I..	J..	R..

Programming a rounding

A rounding is:

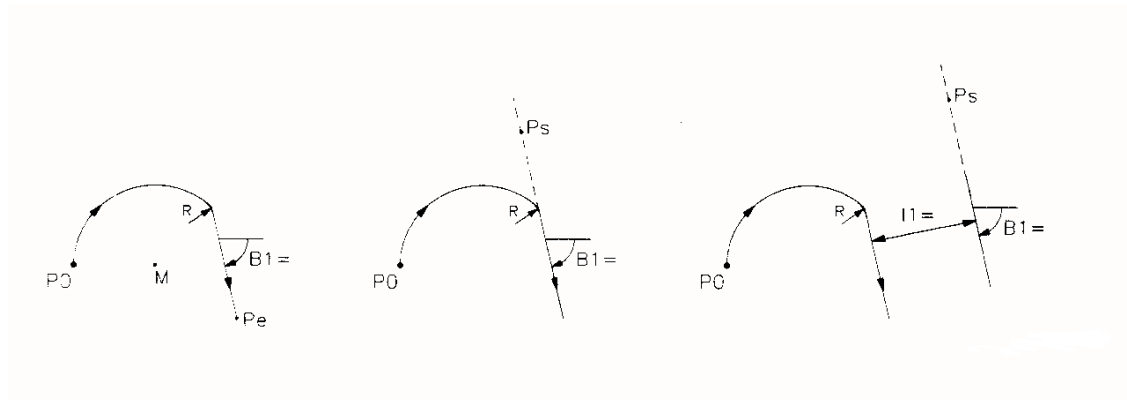
- always tangent to the geometry elements (line or circle) from the previous block and the next one.
- programmed with:
 - G2 or G3 indicating the direction of movement,
 - the radius (R-word) of the rounding.

A rounding between intersecting line - circle or circle - line



Line to circle

N..	G1	{B1=..}	or	{support point}	J1=1/2
N..	G2/G3	R..			
N..	G3/G2	I..	J...	R..	



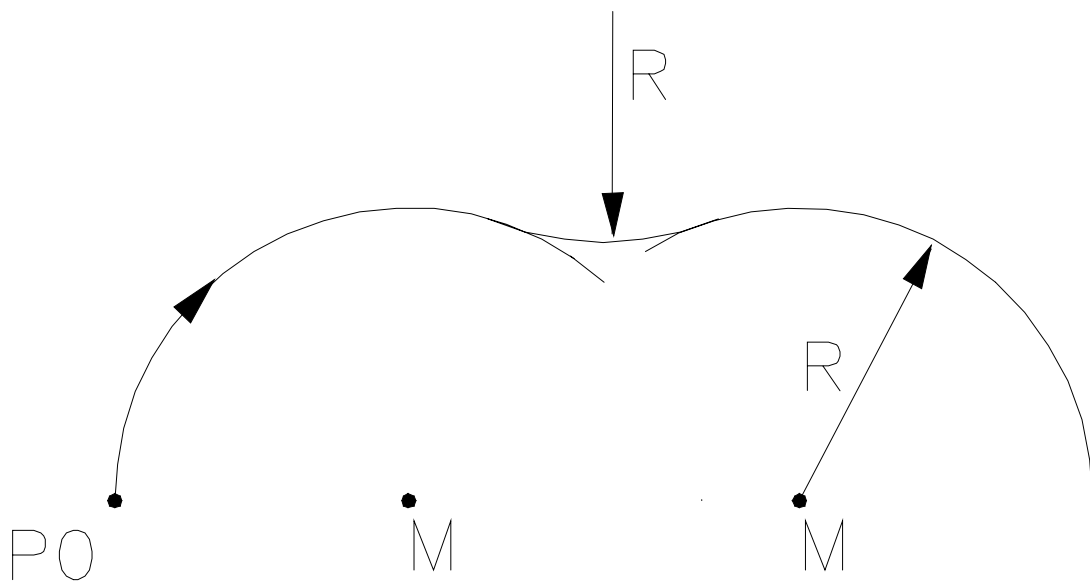
Circle to line

N.. G3/G2 I..... J... {R..} J1=1/2

N.. G2/G3 R..

N.. G1 B1=.. {support point} or {endpoint}

A rounding between two intersecting circles



N.. G2/G3 I.. J.. {R..} J1=1/2

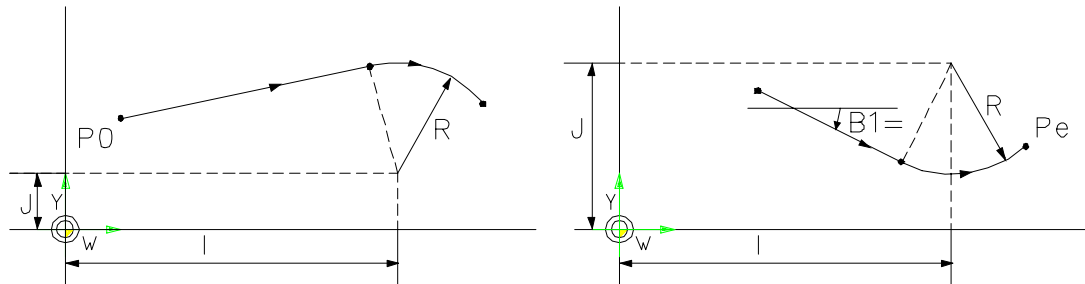
N.. G3/G2 R..

N.. G2/G3 I.. J.. R..

5.32.4 Two tangent geometry elements

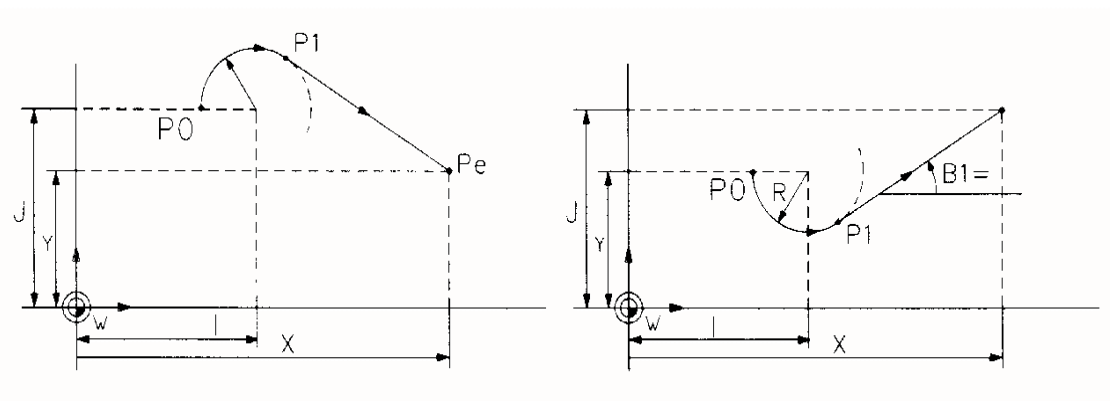
5.32.4.1 Tangency indicator

With the word R1=0 in the first block is indicated, that a line is tangent to a circle or a circle tangent to a line or another circle.



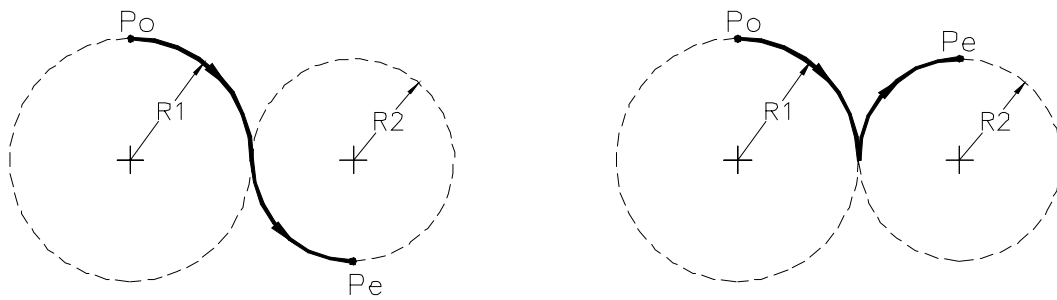
Line tangent to circle

N.. G1 {B1=..} or {support point} R1=0
 N.. G2/G3 I.. J.. R..



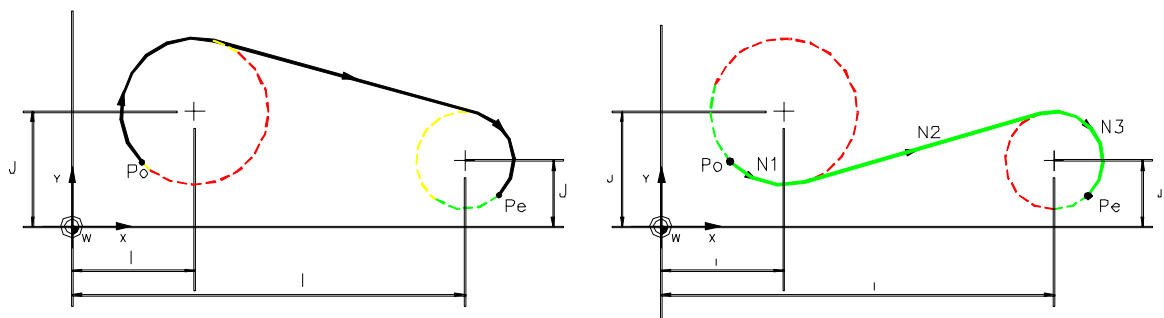
Circle tangent to line

N.. G2/G3 I.. J.. (R..) R1=0
 N.. G1 {B1=..} or {support point} or {end point}



Two tangent circles

N.. G2/G3 I.. J.. {R..} R1=0
N.. G2/G3 I.. J.. R..



Common tangent line from two circles

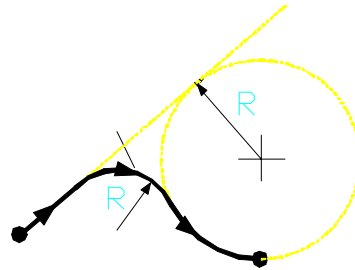
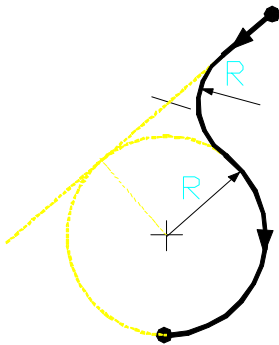
N.. G2/G3 I.. J.. {R..} R1=0
N.. G1 R1=0
N.. G2/G3 I.. J.. R..

Connecting circles

A connecting circle is:

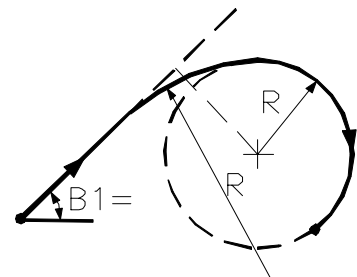
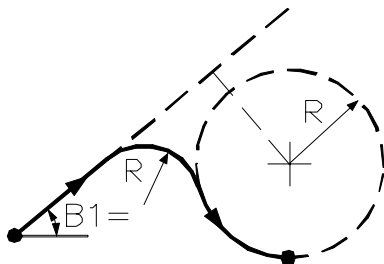
- always tangent to the geometry elements (line or circle) from the previous block and the next one.
- programmed with:
 - G2 or G3 indicating the direction of movement,
 - the radius (R-word) of the circle.

5.32.4.2 A connecting circle between line and circle or circle and line



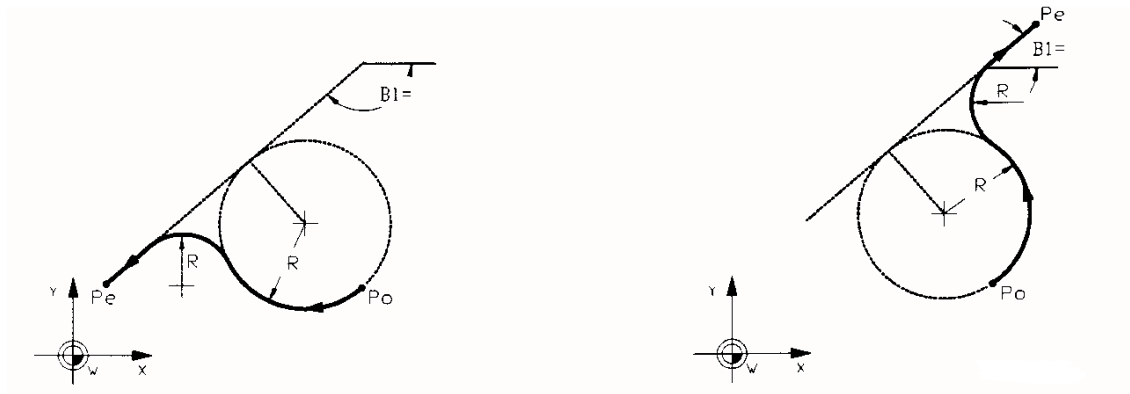
Line tangent to circle

N.. G1 {B1=..} or {support point} R1=0
 N.. G3/G2 R..
 N.. G2/G3 I.. J.. R..



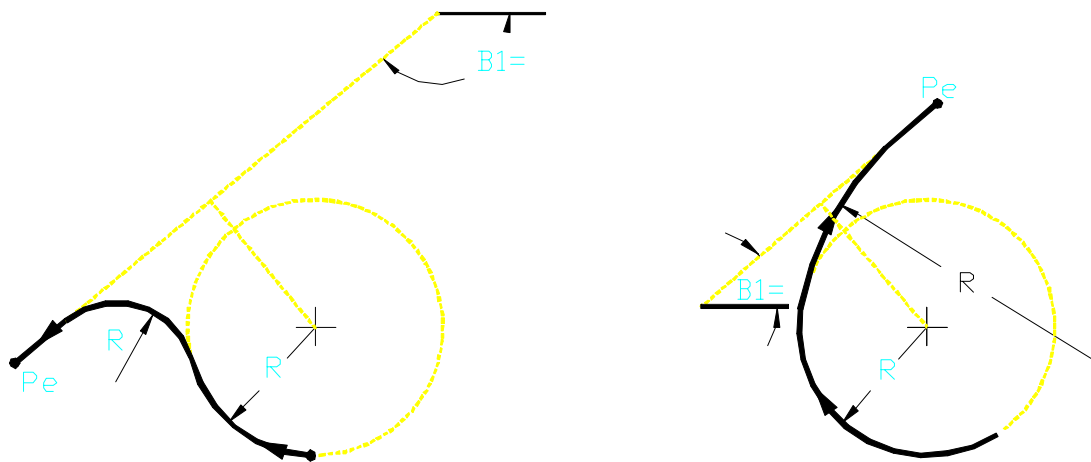
Line does not meet a circle

N.. G1 {B1=..} or {support point}
 N.. G3/G2 R..
 N.. G2/G3 I.. J.. R..



Circle tangent to line

N.. G2/G3 I.. J.. R.. R1=0
 N.. G3/G2 R..
 N.. G1 {B1=..} or {support point} or {end point}

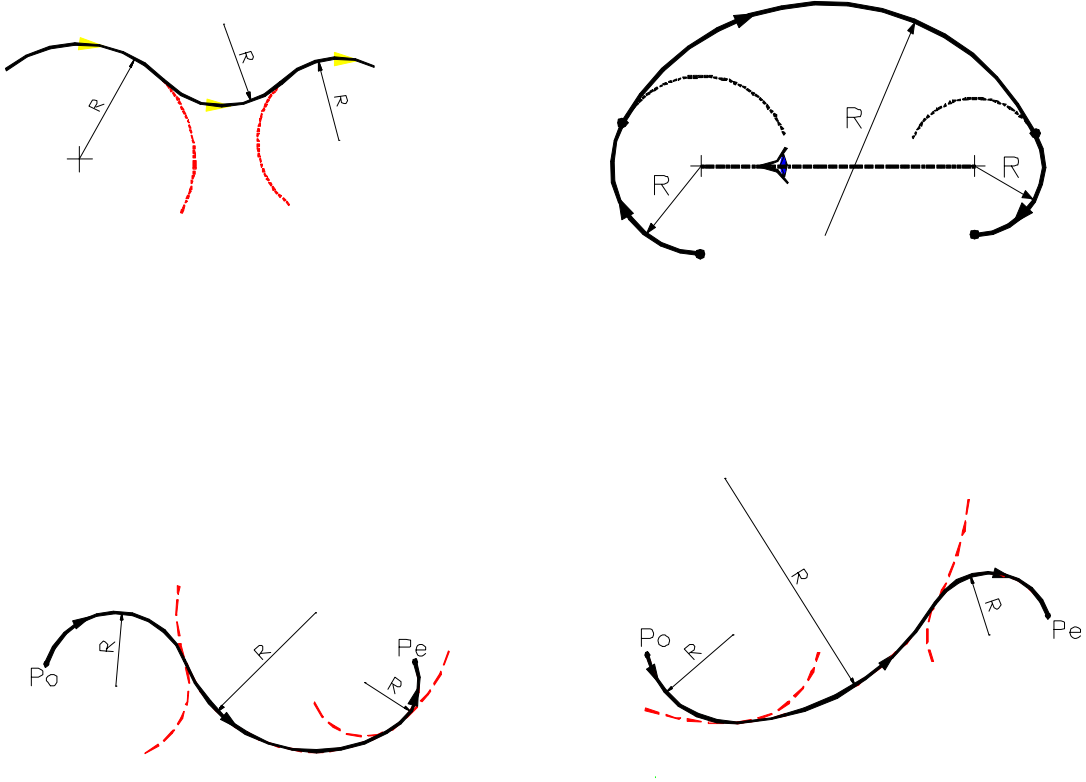


Circle does not meet the line

N.. G2/G3 I.. J.. R..
 N.. G3/G2 R..
 N.. G1 {B1=..} or {support point} or {end point}

5.32.4.3 A connecting circle between two circles outside each other

To insert a connecting circle between two circles outside each other which do not meet. The direction of rotation on the three circles indicates the type of connecting circle.

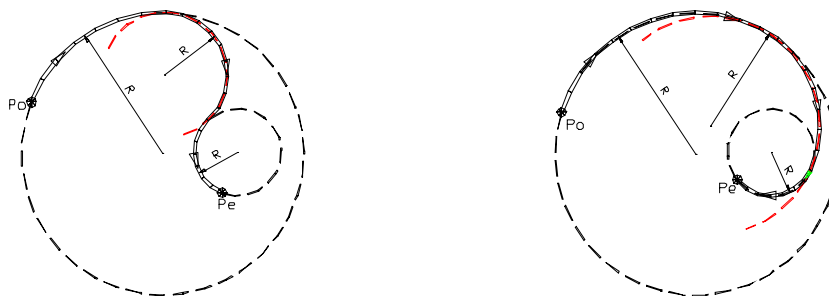


For all cases the same format is available:

N..	G2/G3	I..	J..	{R..}
N..	G3/G2	R..		
N..	G2/G3	I..	J..	R..

5.32.4.4 A connecting circle between two circles of which one circle inside the other one

To insert a connecting circle between a circle inside the other one which do not meet. The direction of rotation on the three circles indicates the type of connecting circle.



For both cases the same format is available:

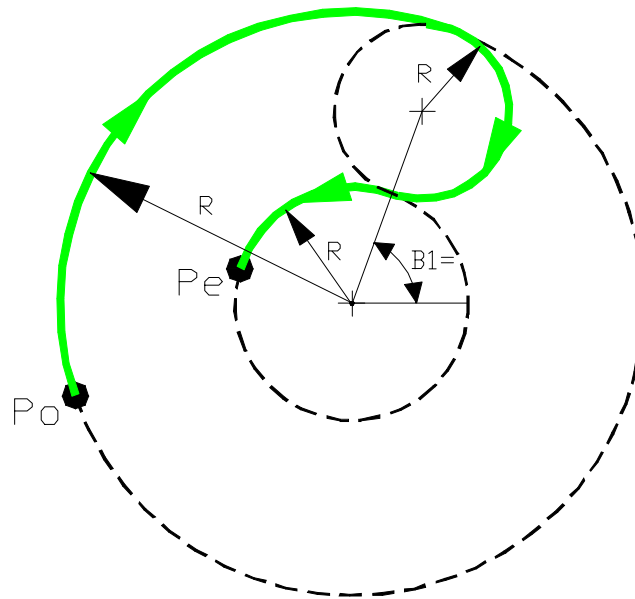
N..	G2/G3	I..	J..	{R..}
N..	G2/G3	R..		
N..	G2/G3	I..	J..	R..

5.32.4.5

A connecting circle between two concentric circles

Two concentric circles are a very special case of one circle inside the other one. In this case the centre points of both circles coincide. The word B1=.. which indicate the angle with the main axis of the line through the centre point of the concentric circles and the connecting circle, is used as additional information and has to be inserted in the block with the connecting circle.

The formats are:

Radius of the connecting circle is known

Two concentric circles

N..	G2/G3	I..	J..	{R..}
N..	G2/G3	R..	B1=..	
N..	G2/G3	I..	J..	

Radius of the second circle is known

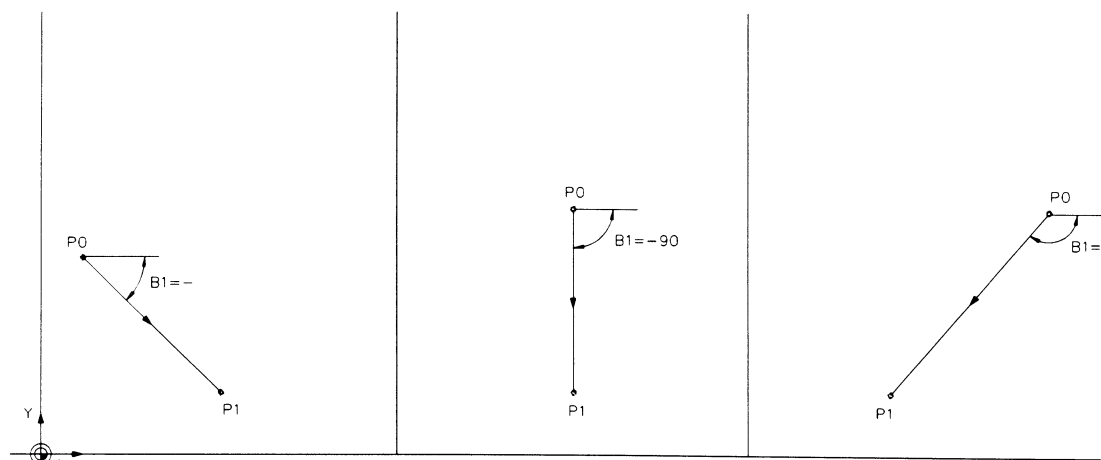
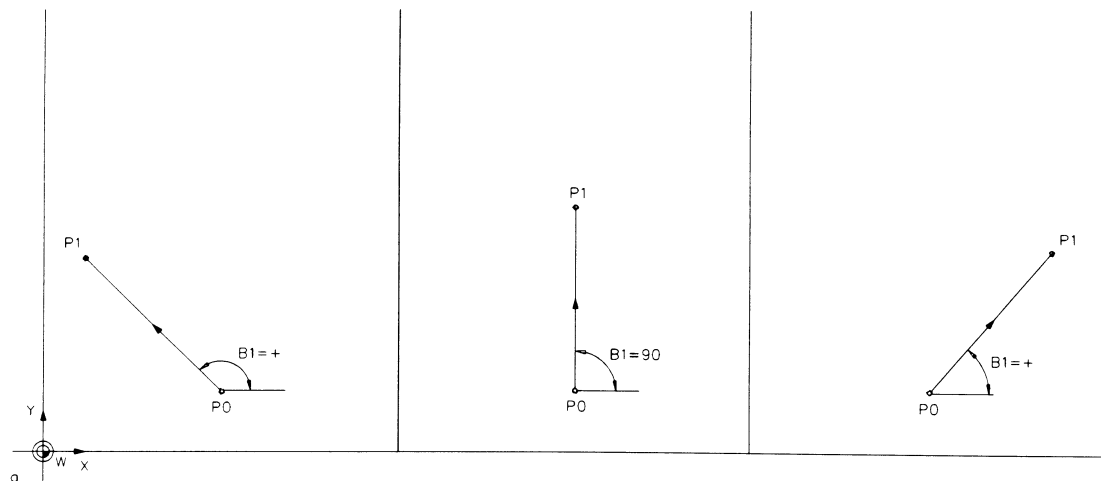
In this case the radius of the connecting circle will be calculated.

N..	G2/G3	I..	J..	{R..}
N..	G2/G3	B1=..		
N..	G2/G3	I..	J..	R..

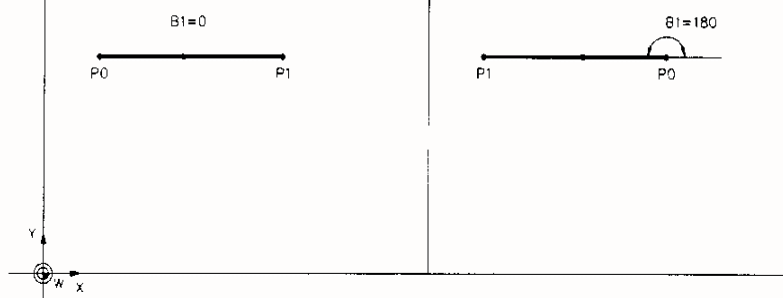
5.32.5 Line definitions

Programming the angle b1=

In a lot of cases a linear movement has to be programmed with the angle, which the line makes with the main axis. The angle is programmed with the word B1=.. The angle should be programmed in the direction of movement, which means that one should look from the start point of the first movement to the end point of it. The sign of the angle contains the direction of movement and can be seen from the illustration.



391



Note: It is important that the angle is programmed correctly, otherwise the wrong intersection point can be chosen.

END POINT An end point is programmed with:

- the absolute Cartesian coordinates X and Y
- the polar coordinates B2= and L2=
- a previously defined point P or P1=

In some cases the intersection point of two elements is known from the drawing and can be programmed as an end point. It is still possible to insert a chamfer between linear movements or a rounding between intersecting elements.

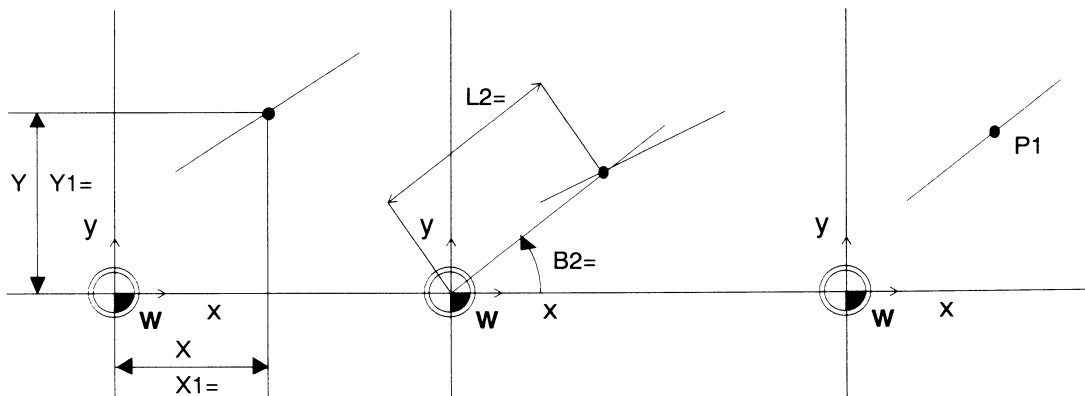
If the intersection point of two lines is programmed as an end point, this point is assumed to be the start point of the next movement and can be programmed with one (X.. or Y..) two coordinates (X.. and Y..) one coordinate and angle (X.. or Y.. and B1=..).

A horizontal or vertical line can also be programmed with one coordinate only. The other coordinate is picked up from the previous blocks. If the start point of the first element is not known, the angle which the line makes with the main axis, has to be added to the block.

CIRCLE CENTRE POINT

The circle centre is programmed with either the absolute Cartesian coordinates (I, J, K..) or its polar coordinates (B3=, L3=..).

SUPPORT POINT When end point coordinates are unknown, another point on the same line can be used to support the calculations of the end point.



NB8010

X.. Y.. I1=0 B2= L2= I1=0 P1 I1=0 X1=.. Y1=
Defining a support point.

Four formats are available:

N..	G1	{B1=..}	X1=..	Y1=..	
N..	G1	{B1=..}	X..	Y..	I1=0
N..	G1	{B1=..}	B2=..	L2=..	I1=0
N..	G1	{B1=..}	P..		I1=0

Any point on the line can be used as a support point. Its coordinates can be programmed with:

- the absolute Cartesian coordinates X1=, Y1= or X, Y
- the polar coordinates B2=, L2=
- a previously defined point P or P1=

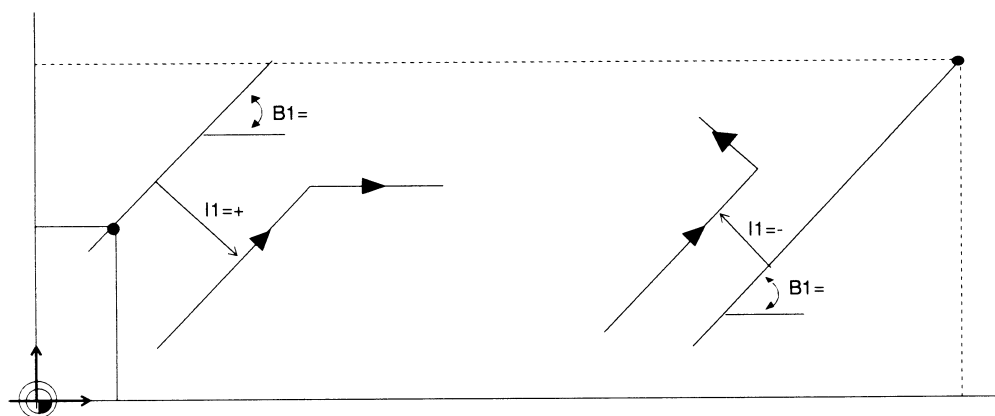
- Note:
1. If the end point coordinates can not be used due to the effect that the tool moves to the end point, a support point with I1=0 has to be programmed.
 2. If a support point is programmed and the line is not yet completely defined, the angle B1=.. which the line makes with the main axis, has to be programmed too. If a block contains too much information, an error message is displayed.

5.32.5.1 Parallel line

Sometimes a line is drawn parallel to a known line. The distance between the required line and the known one is programmed with the word I1=. The word I1= has a sign:

I1=+..: the line to the **right** of the existing line

I1=-..: the line to the **left** of the existing line



NB9022

Defining a parallel line

The following formats are available:

N..	G1	B1=..	X..	Y..	I1=+/-..
N..	G1	B1=..	B2=..	L2=..	I1=+/-..
N..	G1	B1=..	P..	P1=..	I1=+/-..

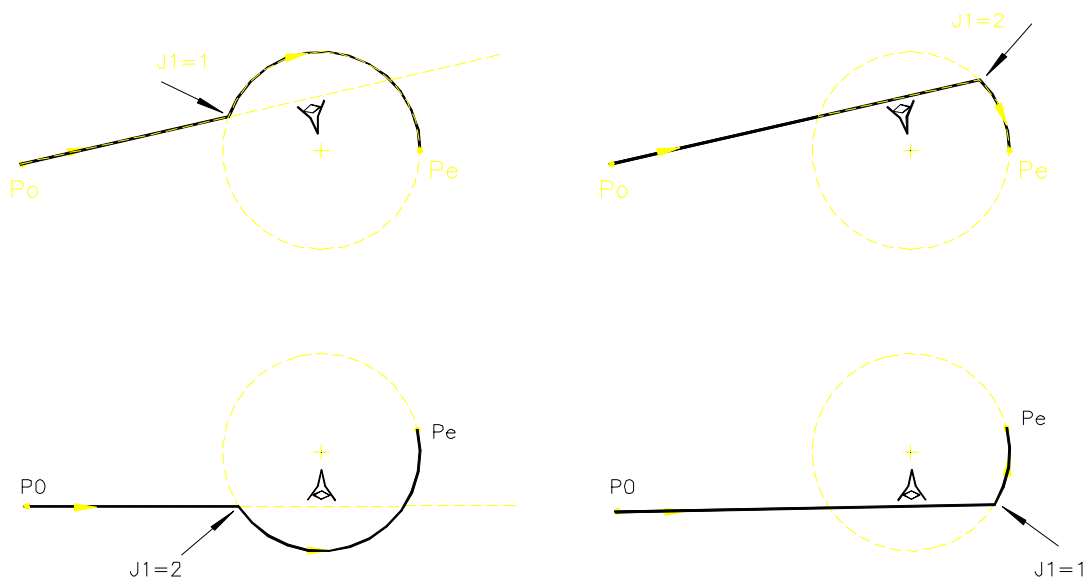
Any point on the existing line can be used. Its coordinates can be programmed with:

- the absolute Cartesian coordinates X, Y
- the polar coordinates B2=, L2=
- a previously defined point P or P1=

5.32.5.2 Intersection point indicator

When a line or circle or two circles cross each other, there will be two possible points of intersection. A special word (J1=1 or 2) is used to indicate which intersection point's coordinates must be calculated. Two main methods have to be used for determining which intersection point belongs to J1=1 and which one to J1=2.

1. when the line goes past the circle's centre

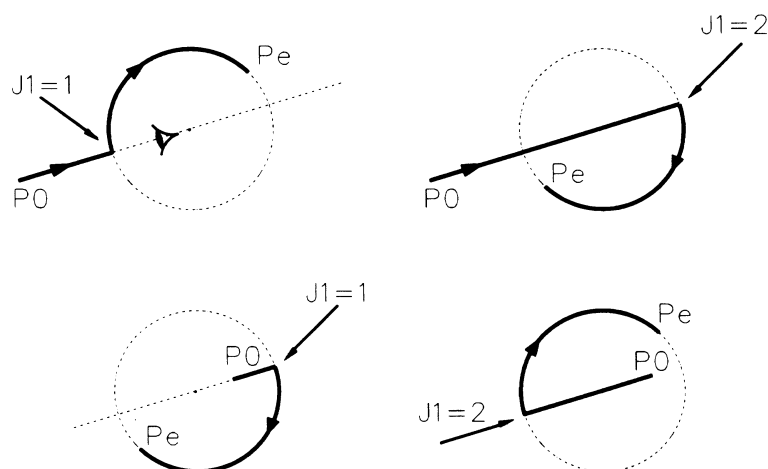


From the centre of the circle look at the line. The J1=1 intersection will be on the left and J1=2 intersection on the right of the perpendicular.

2. when the line goes through the centre of the circle

2.1 line intersects circle

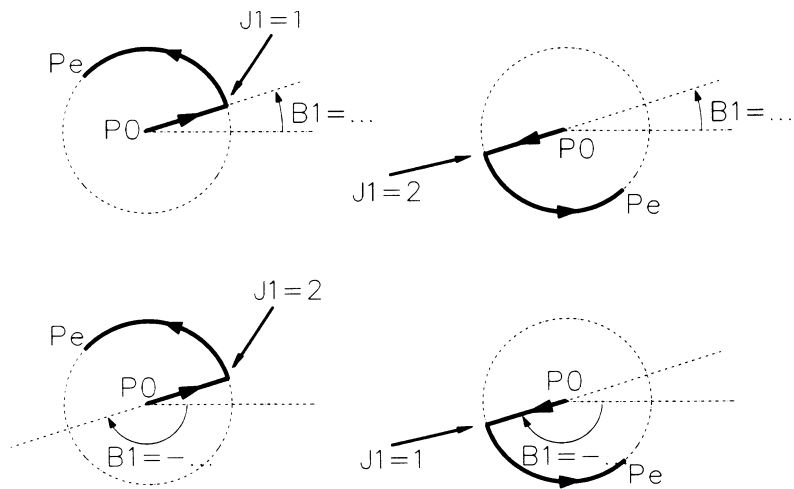
a. start point of the line is not in the circle centre



NB9028

The intersection point closest to the start point is $J1=1$; the other point is $J1=2$

b. start point of line is in the circle centre

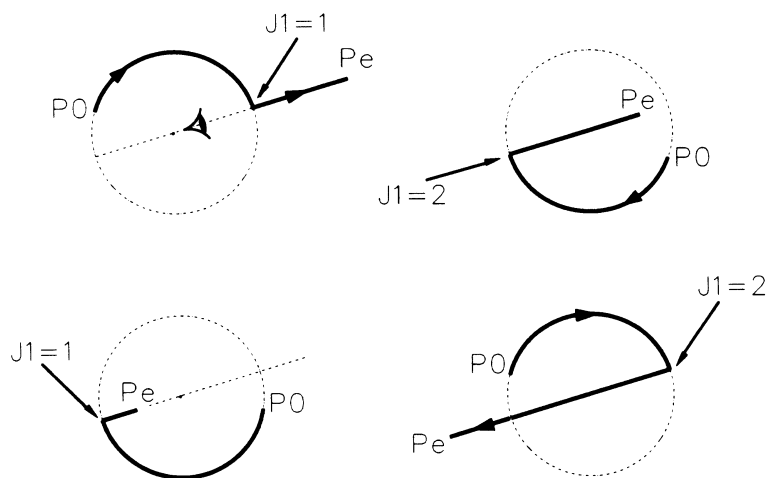


NB9030

The intersection point in the direction of movement on the line defines $J1=1$; the other point is $J1=2$.

2.2 circle intersects line

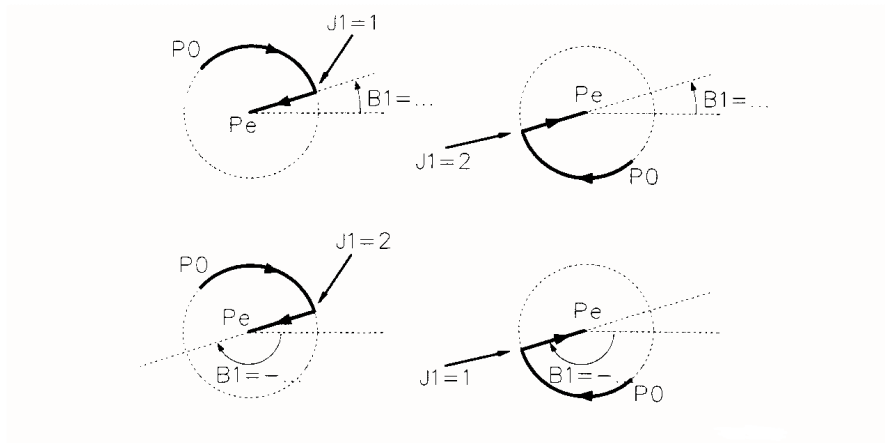
a. end point of the line is not in the circle centre



NB9029

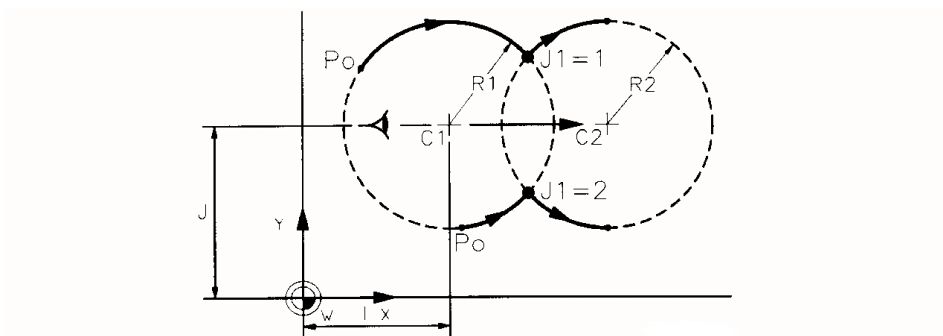
The intersection point closest to the end point is $J1=1$; the other point is $J1=2$

b. end point of line is in the circle centre



The intersection point in the direction of movement on the circle defines $J1=1$; the other point is $J1=2$.

3. two circles intersect each other



When two circles intersect each other, the left ($J1=1$) or right ($J1=2$) intersection point is determined by looking from the centre point of the first circle to the centre point of the second one and seeing which intersection point is on the left or on the right from the line through the centres.

Note: Refer to PROGRAMMING THE ANGLE $B1=$ for the meaning of in the direction of movement on the line or circle.

Notes and usage

CONTINUOUS AND NON-CONTINUOUS MOVEMENT

With a **continuous** movement the tool moves always in the forward direction. If more than one connecting circle is possible, it depends on the direction of movement on both elements, which connecting circle is automatically taken by the control as default circle. If a short or long arc with a connecting circle is possible, the shorter arc is chosen.

With a **non-continuous** movement:

- the tool can move backwards
- the toolpath can intersect itself
- the longer are with a circular movement can be taken.

In some milling applications the non-continuous movements have to be used e.g. if the longer are of a connecting circle should be programmed. In applications like laser cutting the non-continuous movements can be very useful.

Refer to the appendix for programming non-continuous movements.

CANCELLATION

The geometric calculations are cancelled with the function G63. Thereafter complete blocks have to be programmed. In the last block before the cancellation of the geometric calculations an absolute position must be programmed.

DEFAULT MODE

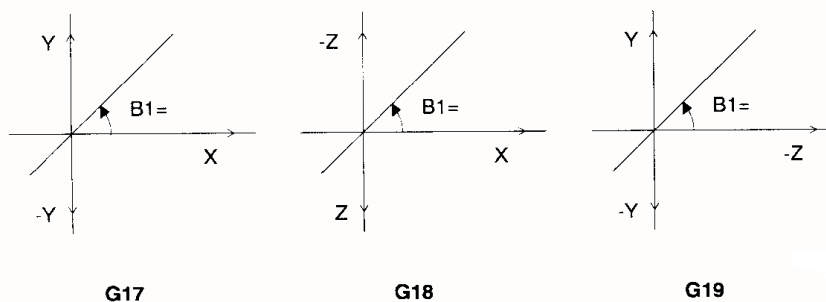
At CLEAR CONTROL the function G63 is automatically activated.

PLANE SELECTION

The geometric calculations are executed in the plane defined by G17 (XY- plane), G18 (XZ-plane) or G19 (YZ-plane).

In the three planes the angle B1= is defined with respect to:

- the + X-axis in the XY- or XZ-plane
- the - Z-axis in the YZ-plane



Angle definition in the different planes

A support point can be programmed with:

- X1= and Z1= in the XZ-plane
- Y1= and Z1= in the YZ-plane

PERMITTED FUNCTIONS

G-functions allowed when the G64-function is active:

G0/G1/G2/G3; G4; G25/G26; G27/G28; G40/G41/G42/G43/G44; G94/G95

Functions **not allowed** when G64 is active

- All G-functions not mentioned in the table above
- Incremental programming (Cartesian and polar)
- Helix interpolation
- More than one defined point in a block
- The M-functions M6, M66 and M67

USING MACROS

The use of the geometric calculations is allowed in a macro. All geometry blocks including G63 and G64 must be in the same macro.

USING REPEAT FUNCTIONS

The use of the geometric calculations is allowed in a section of a partprogram repeated by a G14 or G29. All geometry blocks including G63 and G64 must be in the same section to be repeated.

SCALING, MIRROR IMAGE AND AXES ROTATION

First activating scaling, mirror image or axes rotation and then using the geometric calculations is allowed and results in the required operation on the program section.

Refer to chapter 13 "Geometric calculations with continuous movements" on page 597 .

5.33 G66/G67 Select negative/positive tool direction

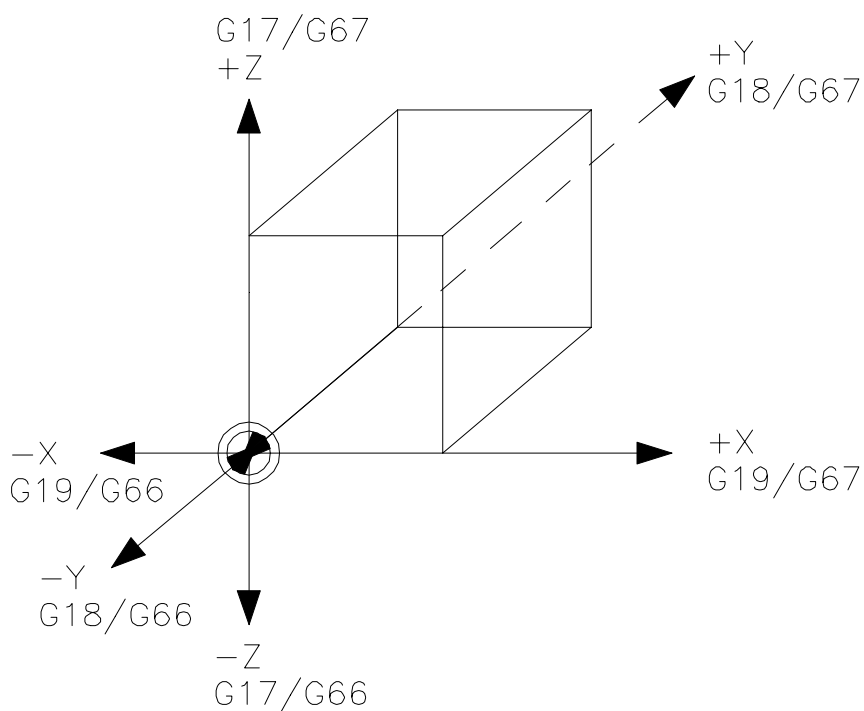
Purpose

To select the direction in which the tool is pointing:
Tool length compensation in - direction/+ direction G66/G67.

G66: Tool is pointing in the negative direction of the tool axis

G67: Tool is pointing in the positive direction of the tool axis

The use of these functions allows a user always to enter a positive tool length value into the tool memory and the programmer to always look from the tool at the plane for circular interpolation and radius compensation.



Format

Tool pointing in the negative direction of the tool axis:

N... G66

Tool pointing in the positive direction of the tool axis:

N... G67

Parameter

None

Type of function

Modal

Associated Functions

G17/G18/G19

Notes and usage**DEFAULT MODE**

The function G66 is automatically activated when the CNC is switched on, thus for a tool pointing in the negative direction along the tool axis.

AVAILABILITY

This G66/G67 function is not active in all versions, because it cannot be used on all machine types. It may also be that this function is only possible in G19.

TOOL LENGTH IN THE TOOL MEMORY

The tool length stored in the tool memory is always a positive value, unless corrections on the length of a standard tool are processed.

TOOL LENGTH COMPENSATION

With G66 active (default mode) the tool length compensation is performed in the negative direction of the tool axis.

With G67 active the tool length compensation is performed in the positive direction of the tool axis.

CIRCULAR INTERPOLATION

To determine the direction of rotation on a circular the partprogrammer looks in the negative (G66) or positive (G67) direction of the tool axis at the plane in which the circle is made. In both cases G2 is used for a clockwise movement and G3 for a counter clockwise movement.

With G67 active the CNC makes the necessary conversions automatically during program execution.

RADIUS COMPENSATION

To determine if the tool is moving on the left or on the right of the workpiece the partprogrammer looks in the negative (G66) or positive (G67) direction of the tool axis at the plane in which radius compensation is made. In both cases G41 is used for the tool moving on the left and G42 for the tool moving on the right of the workpiece.

With G67 active the CNC makes the necessary conversions automatically during program execution.

FIXED CYCLES

If the tool is pointing in the positive direction of the tool axis, the depth of the fixed cycle must be programmed with a positive sign (+) to indicate that the cycle is to be executed in the positive direction of the tool axis. The sign is not automatically inverted.

With the milling cycles (G87 to G89):

- The direction of rotation on the circular arcs is automatically changed in the opposite direction
- The milling direction programmed with the J-word, is not automatically changed.

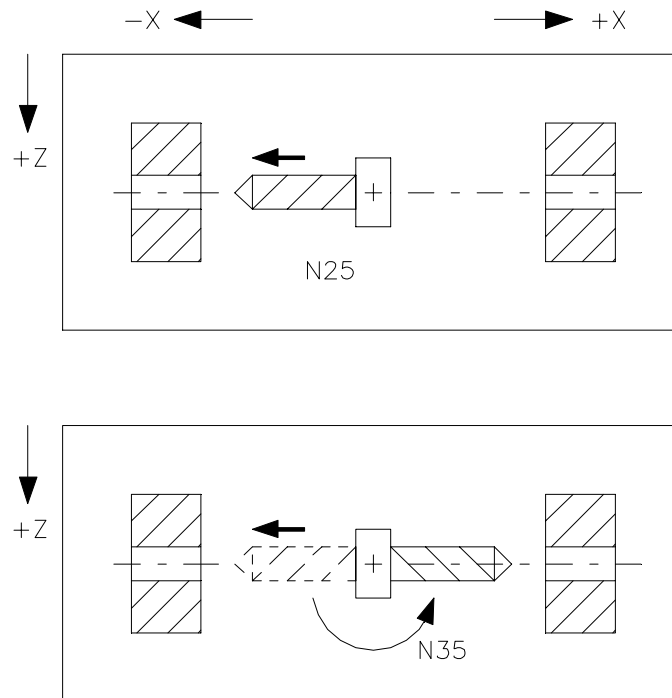
CANCELLATION

Both functions cancel each other. CLEAR CONTROL, by M30 or by Softkey CANCEL PROGRAM, does not cancel them.

RESTRICTION

Do not use the geometry (G64 active) in combination with G67. No error message is displayed, if the geometry is activated after G64. Contour errors may occur, because not all geometry functions are converted properly.

Example



Upper picture G66 active
Lower picture G67 active

N25 G1 [End point coordinates]

N30 G67

N35 G1 [End point coordinates]

Explanation

N25: First hole is drilled.

N30: Select tool to point in the positive direction of the tool axis.

N35: Second hole is drilled.

5.34 G70/G71 Inch/Metric programming

Purpose

Allows the loading of partprograms, which use a different dimensional unit system from that, currently active in the CNC.

G70: dimensional units of the partprogram are in inches.

G71: dimensional units of the partprogram are in millimetres.

Format

Inch programming:

N... (PROG. NAME) G70

Metric programming:

N... (PROG. NAME) G71

Parameter

None

Type of function

Modal.

Notes and usage

DIMENSIONAL UNITS

Units for linear dimensions:	.001 mm	.0001 inch
------------------------------	---------	------------

Units for feed rate	(G94):	.001 mm/min	.0001 inch/min
	(G95):	.001 mm/rev	.0001 inch/rev

Units for cutting speed:	1 m/min	1 feet/min
--------------------------	---------	------------

Note: The cutting speed is used in the technology tables.

ACTIVE SYSTEM OF DIMENSIONAL UNITS

With a machine constant (MC707) is determined which type of dimensional units is used automatically by the CNC at initialising. This is the active system of dimensional units.

Note: The functions G70 and G71 are used at the input level of programmed data. They do not influence the measuring system on the machine tool.

CHANGING THE SYSTEM OF DIMENSIONAL UNITS

If the active system of dimensional units has to be changed, e.g. if a partprogram in the other unit system should be entered via the keyboard, the machine constant setting has to be changed and the control reinitialised.

After initialising all dimensions in the memories are divided by 10 (changing from metric to inch) or multiplied by 10 (from inch to metric). Refer also to CNC MEMORIES.

UNIT CONVERSION AT LOADING A PROGRAM FROM A DATA CARRIER

If the CNC detects a G70 or G71 during the loading of a program from a data carrier, the CNC checks if the units used in the program and the active dimensional unit system are the same. If a difference is detected, the CNC converts the coordinates of the linear axes and the feedrates into the equivalents of the active system e.g. 'X1' (1 inch) is converted into 'X25.4' (25.4 mm). Also the function G70 or G71 is changed automatically by the control to the opposite function.

Note: Only one type of dimensional units is permitted in a program.

EXECUTING A PARTPROGRAM

If a G70 or G71 is not programmed at the beginning of a partprogram, the CNC assumes that all dimensions are in accordance with the unit system activated on the control.

If one of the functions G70 or G71 is programmed, the CNC checks if the stored program is in the same units as the active system of the control. If a difference occurs, an error is generated.

CNC MEMORIES

The CNC memories in which the tool dimensions, zero offsets, defined points and technology values are stored, must always be in the units of the active dimensional system. If this system is changed, all the stored values must be re-entered to their equivalents in the new unit system. The parameter memory is not influenced by a change to the other unit system.

ENTERING A PROGRAM VIA THE KEYBOARD OF THE CONTROL

Programs, which are entered into the memory via the keyboard of the control, cannot contain a G70 or G71, which conflicts with the active dimensional system. If this is detected, an error message is generated.

Examples

EXAMPLE 1

CNC active system of unit's - metric
Partprogram values are in inches.

N9001 (EX.1) G70

N50 G1 X2 Y1.5 F8

Reading block N50 into the partprogram memory results in storing the coordinates X50.8 Y38.1 and a feed rate of 203.2 mm/min.

EXAMPLE 2

CNC active system of unit's - inches.
Partprogram values are in millimetres.

N9002 G71

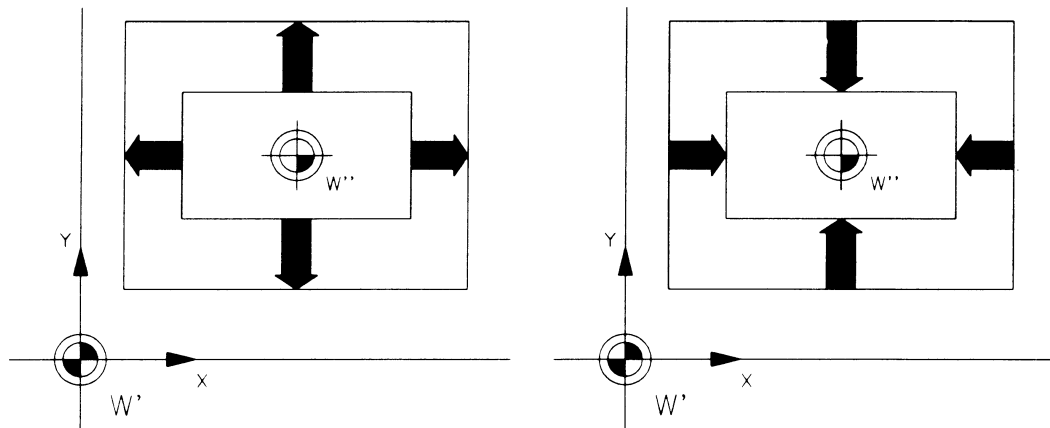
N50 G1 X50.8 Z38.1 F203.2

Reading block N50 into the partprogram memory results in storing the coordinates X2. Y1.5 and a feed rate of 8 inches/min.

5.35 G72/G73 Cancel/Activate scaling or mirror imaging

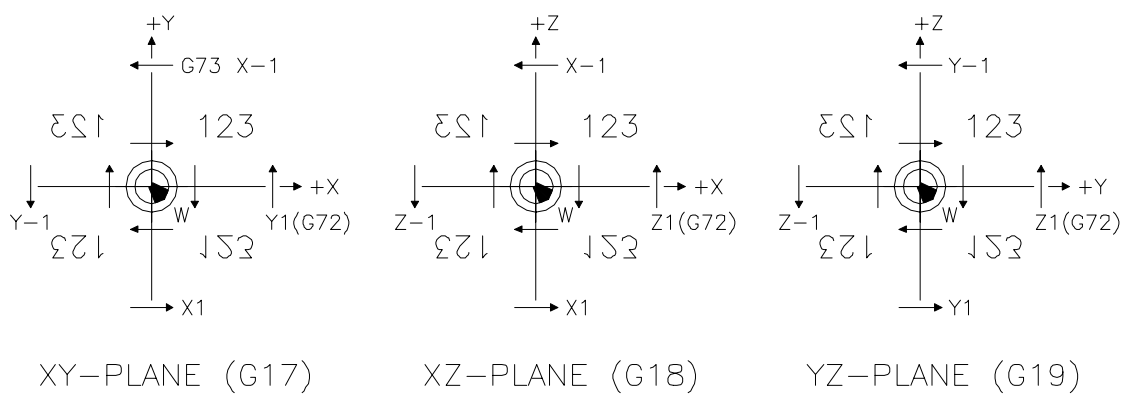
Purpose

1. To scale (enlarge or reduce in shape) a group of axis coordinates.



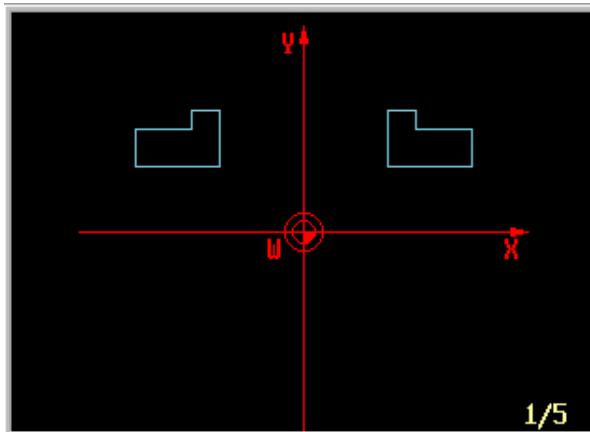
NB8684

2. To produce a mirror image of a group of linear main axis coordinates or a change of sign of rotary axis coordinates. (sign inversion)



- G72: To cancel scaling and mirror imaging.
 G73: Activate scaling and/or mirror imaging.

G73:

**Format**

To activate scaling

N... G73 A4=...

To cancel scaling

N... G73 A4=1 (factor) or A4=100 (percentage)

To produce a mirror image around an axis or a sign inversion of the axis.

N... G73 {X-1} {Y-1} {Z-1} {A-1} {B-1} {C-1}

To cancel mirror image / sign inversion per axis

N... G73 {X1} {Y1} {Z1} {A1} {B1} {C1}

To cancel scaling and mirror image

N... G72

Parameters

G72: No special words

Modal words

H, T1=

Parameter G73

```

G   Mirror image and scaling
X   X-1:set mirror image / X1:reset
Y   Y-1:set mirror image / Y1:reset
Z   Z-1:set mirror image / Z1:reset
A   A-1:set mirror image / A1:reset
B   B-1:set mirror image / B1:reset
A4= Scaling factor

```


Modal words

F, S, H, E..=

T1=, T2=,

Type of function

Modal.

Associated functions

G92/G93 axis rotation

Notes and usage

SCALING

SCALING PARAMETER A4=

The machine constant MC714 and MC715 determine if the A4= parameter is a factor (format 2.6) or a percentage (format 3.4).

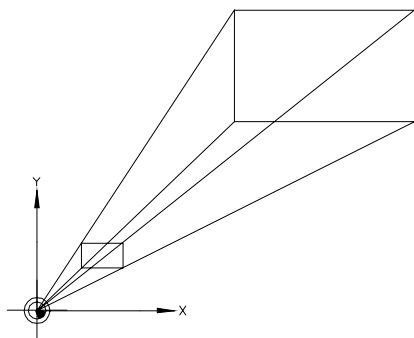
The format of the factor is set with another machine constant.

So a dimension increase of 1.25% is programmed as:

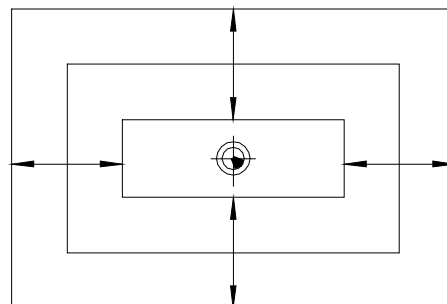
- A factor: G73 A4=1.0125

- A percentage: G73 A4=101.25

GEOMETRIC CENTRE OF THE GROUP OF COORDINATES



Scaling about zero point W



Scaling about geometric centre

The scaling function uses the current zero point W as the starting point. If necessary, this point should be set by the use of a G92/G93 zero point shift at the geometric centre of the group of axis coordinates before the scaling operation. This ensures that the coordinates are symmetrically scaled around a fixed point, which is not moved out of position by the scaling operation.

PROGRAMMED ZERO POINT SHIFTS (G92/G93)

G92/G93 zero point shifts are scaled if they are present in a group of coordinates to be scaled.

PROGRAMMED ZERO POINT SHIFTS (G92/G93)

G92/G93 zero point shifts are scaled if they are present in a group of coordinates to be scaled.

G51-G59 ZERO POINT SHIFTS

The G51-G59 zero point shifts are not influenced by the scaling operation.

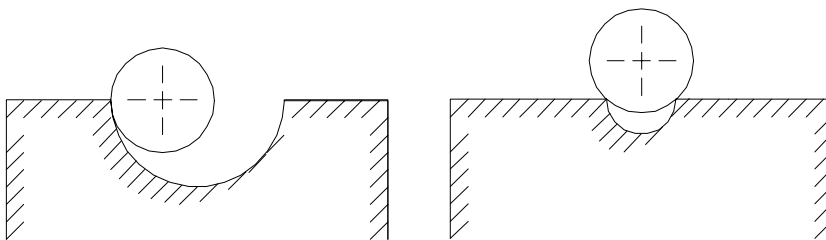
SCALING THE TOOL AXIS

With the machine constant for the factor is also determined if scaling is applied to only the axes coordinates in the main plane or to the tool axis as well.

TOOL DIMENSIONS

If the tool axis is to be scaled, the tool length is not scaled.

Tool diameters are not scaled.



Before scaling

After scaling - tool is too large

When scaling is to be performed, the programmer must decide if the existing tool diameter is suitable for the different dimensions.

CANCEL SCALING

The scaling is cancelled by:

- G72 mirror image if active, is cancelled too
- Softkey CLEAR CONTROL, M30 and Softkey CANCEL PROGRAM, both scaling and mirror image are cancelled
- G73 and the scaling factor: A4=1 or A4=100.

MIRROR IMAGE

SIGN INVERSION

Mirroring around the Y-axis in e.g. the XY-plane means changing the sign of the X-coordinate in the opposite sign, thus +X to -X and vice versa (sign inversion).

A rotary axis cannot be mirrored around an axis, but a sign inversion is still possible, thus +B to -B.

PLANE SELECTION

Mirroring has a meaning in the main plane only

G17 active:

X-1: mirroring around Y-axis

Y-1: mirroring around X-axis

Z-1: sign inversion in tool axis

G18 active:

X-1: mirroring around Z-axis

Z-1: mirroring around X-axis

Y-1: sign inversion in tool axis

G19 active:

Y-1: mirroring around Z-axis

Z-1: mirroring around Y-axis

X-1: sign inversion in tool axis

MIRRORING CIRCULAR MOVEMENTS

When a circular movement is mirrored in one axis, its direction of rotation is also reversed: G2 becomes G3 and G3 becomes G2. This ensures that the tool travels in the correct direction when moving on circular arcs.

TOOL RADIUS COMPENSATION

Tool radius compensation is automatically reversed when mirroring occurs in one axis, for example: G41 becomes G42. This ensures that the tool radius compensation is correctly calculated from the programmed coordinates.

PROGRAMMED ZERO POINT SHIFTS (G92/G93)

G92/G93 zero point shifts are mirrored too, if they are present in a group of coordinates to be mirrored.

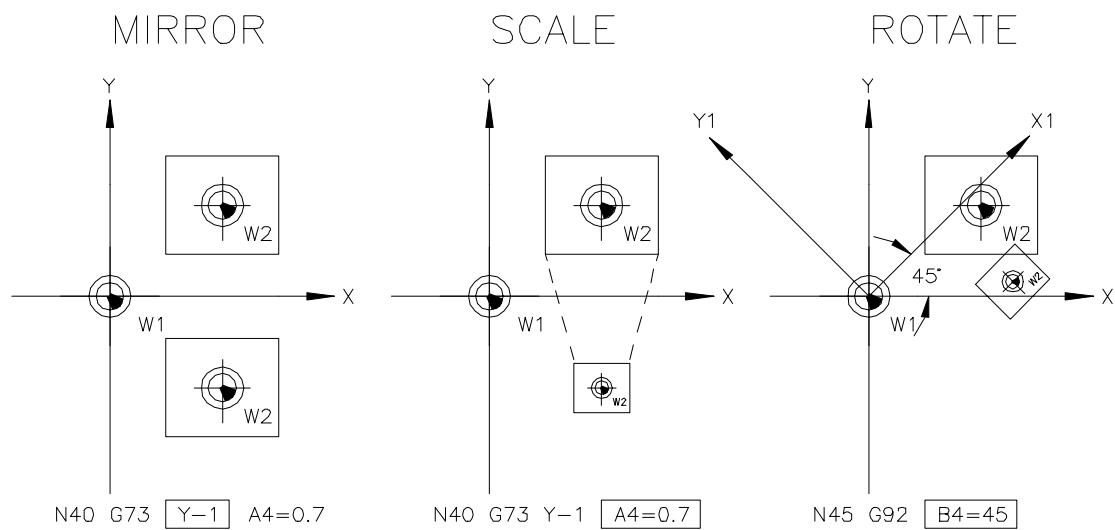
G51-G59 ZERO POINT SHIFTS

The G51-G59 zero point shifts are not influenced by the mirror operation.

SPINDLE ROTATION

The direction of spindle rotation is not reversed by the mirroring operation. The programmer must therefore consider this fact when deciding which axis coordinates are to be mirrored.

SCALING, MIRROR IMAGE AND ROTATION OF AXES



A group of axes coordinates can be scaled, mirrored and rotated by using a combination of the G73 and G92/G93 functions with the word B4=.

Mirror

Scale

Rotate

CANCEL MIRROR IMAGE

Mirror image is cancelled by:

- G72 a scaling operation, if active, is cancelled too
- Softkey CLEAR CONTROL, M30 and softkey CANCEL PROGRAM, both scaling and mirror image are cancelled
- G73 and the positive sign of the mirrored axis;

E.g. X-1 is cancelled by X+1.



N1 G17
N2 G54
N3 S300 T1 M6 (Mill radius 4 mm)
N4 G0 X-5 Y10 Z10 F700 M3
N5 G1 Z-15
N6 G43 Y5
N7 G41
NS G1 X0
N9 G1 Y180
N10 G1 X180
N11 G1 Y0
N12 G1 X5
N13 G1 Y-10
N14 G40
N15 G1 Z10
N16 G73 X-1 Y-1
N17 G14 N1=4 N2=15
N18 G72
N19 S100 T2 M6 (Drill radius 4 mm)
N20 G81 Y10 Z-20 F200 M3

N21 G79 Y60
N22 G79 Y95
N23 G79 Y130
N24 G79 X165
N25 G79 X85
N26 G79 X45
N27 G79 Y60
N28 G73 X-1 Y-1
N29 G14 N1=21 N2=28
N30 G72
N31 G0 Z200 M30

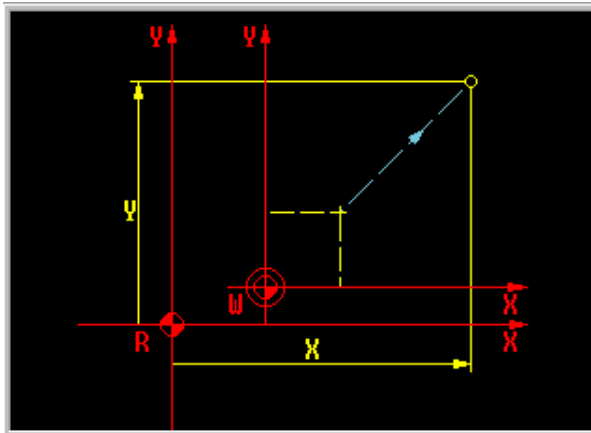
Explanation:

N1: Select XY-plane.
N2: Set program zero point.
N3: Load tool number 1.
N4: Move tool rapidly to programmed position. Set feedrate to 700 mm/min and spindle rotation in clockwise direction.
N5: Feed tool to depth at set feedrate.
N6: Move tool T0 programmed position.
N7: Select tool radius compensation LEFT.
N8-N13: Machine the workpiece.
N14: Cancel tool radius compensation.
N15: Retract tool from workpiece.
N16: Mirror coordinates around X- and Y-axis.
N17: Repeat instructions from block 4 to 15.
N18: Cancel the mirroring operation.
N19: Load tool number 2.
N20: Define drilling cycle and start spindle again.
N21-N27: Execute drilling cycle at programmed points.
N28: Mirror coordinates around X- and Y-axis.
N29: Repeat instructions from block 21 to 27.
N30: Cancel the mirroring operation.
N31: Retract tool in tool axis and end of program.

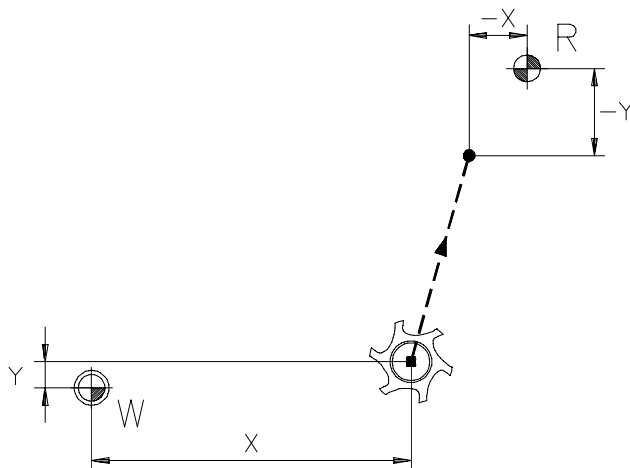
5.36 G74 Absolute position

Purpose

To execute a rapid traverse movement to a position programmed with coordinates measured from the machine reference point R or machine positions.



Format



N... G74 X.. Y.. Z.. {X1=..} {Y1=..} {Z1=..} {K...} {L...} {K2=...}]

Parameter

G	Absolute position
X	Endpoint coordinate
Y	Endpoint coordinate
Z	Endpoint coordinate
A	Endpoint angle
B	Endpoint angle
K	0=inpod / 1=inpos / 2=MC or K2=
L	L0:with toollength / L1:without
X1=	Absolute position MC (1-18)
Y1=	Absolute position MC (1-18)
Z1=	Absolute position MC (1-18)
A1=	Absolute position MC (1-18)
B1=	Absolute position MC (1-18)
K2=	Inpoc window (0: MC, 1-32767 µm)

Modal words
F, S, E...=

Type of function

Non-modal.

Associated functions

G0

Notes and usage

APPLICATION

The G74-function's main application is in programming cycles for tool changers, pallet stations etc., when it is advisable that the programmed coordinates are independent of those used to define the machining of the workpiece.

Note: The absolute position is entered with the addresses X1=., Y1=., K2=.. etc for installation purposes!

END POINT COORDINATES

The end point coordinates can be defined in two different ways:

- 1) X100: relative position to the reference point.
- 2) X100 X1=2: relative position to Home position 2 (MC3146).

For the first axis the machine positions 1 to 10 are describes in machine constants MC3145 – MC3154. For the second axis MC3245 -- MC3254.

When the actual machine constant is zero, no movement will be done.

STOP BETWEEN BLOCKS (K-word)

All programmed axes move simultaneously during the execution of the G74. The next movement starts once all axes have reached their position. There is a stop between the G74-block and the next one as is usual with rapid traverse movements. (K0 is the default setting). With K1 the stop between the blocks can be avoided.

K0: Allowance is made for a (precise) stop between the movement of the G74 block and the movement in the next block, as is usual for rapid traverse movements. (K0 is the start setting).

K1: No allowance is made for a stop between the movement of the G74 block and the movement in the next block (corner rounding). The next movement is started when the desired position is nearly reached in all axes.

K2: No allowance is made for a stop between the movement of the G74 block and the movement in the next block. The next movement is started when the desired position is nearly reached in all axes. This position is defined by machine constant (MC136) (K2=0) or by the window size (K2=...).

K2= Display dimensions in mm (0-32.766 mm)

INCREMENTAL MOVEMENTS

If an incremental movement is programmed after a G74 movement, the coordinates are measured from the position stated in the G74-block.

TOOL LENGTH COMPENSATION

In general tool length compensation is not used with the G74 positioning (L0 is default setting). If length compensation is required, 'L1' must be programmed.

RADIUS COMPENSATION

Radius compensation (G41 - G44) must be cancelled before the G74-function is activated.

GEOMETRY FUNCTION

The G64 geometry function must not be active when G74 is used.

ZERO POINT SHIFTS AND ZERO OFFSETS

The active zero point shift and zero offset are temporarily overridden.

AXIS ROTATION AND SCALING

The programmed G74-position is not affected by axis rotation or scaling.

AFTER EXECUTION A G74 BLOCK

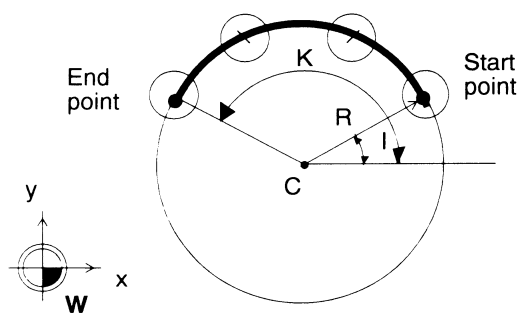
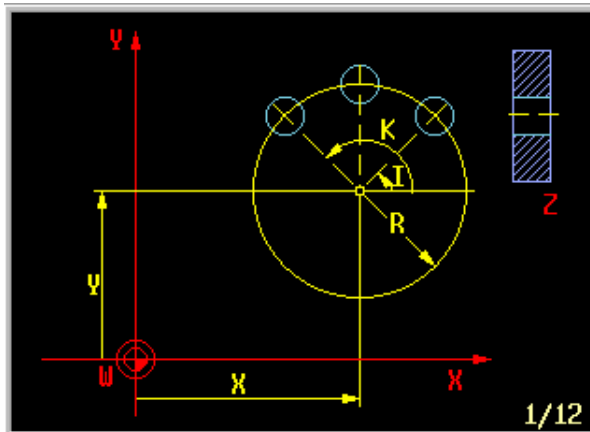
All zero points and the tool length compensation (if suppressed) become active again after a G74-block.

The last movement before G74 is activated must use either the G0 or G1 function. This function is automatically used in the first movement following the G74-block.

5.37 G77 Bolt hole cycle

Purpose

To execute any fixed cycle (G81, G83-G89) at points which are equally spaced on a circular arc or a complete circle.



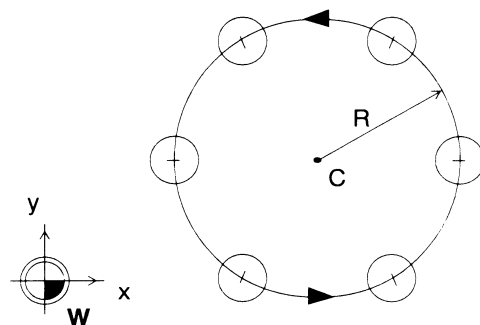
CIRCULAR ARC

Points on an arc

N... G77 [centre point coordinates] R... J... {I...} {K...}.

Points on a complete circle

N... G77 [centre point coordinates] R... J... {I...}



COMPLETE CIRCLE

Parameters

G Bolt hole circle
 X Center point coordinate
 Y Center point coordinate
 Z Center point coordinate
 A Endpoint angle
 B Endpoint angle
 I Angle to first point
 J Number of points
 K Angle to last point
 R Circular pattern radius
 B1= Angle
 B2= Polar angle
 ?90= Centre point abs. (X,Y,Z..)
 ?91= Centre point incr. (X,Y,Z..)

L1= Path length
 L2= Polar length
 P1= Point definition nr.for centre

Type of function

Non-modal

Associated functions

G79, G81, G83-G89

Notes and usage**I ANGLE**

Minimum angle: - 360 degrees, maximum angle: + 360 degrees.

K ANGLE

Minimum angle: 0 degrees, maximum angle: + 360 degrees.

The angle is programmed in degrees and decimal parts of a degree in steps of .001 degrees.

DIRECTION OF EXECUTING THE POINTS

When I-K greater than zero, the holes are in the CW direction.

When I-K smaller than zero, the holes are in the CCW direction.

ROTATED POCKET OR GROOVE (B1=)

A previously defined pocket or groove (G87 or G88) can be rotated about an angle. The centre of rotation is the point used in the G77 block to program the location of the pocket or groove.

The angle is programmed with the word B1= in degrees and decimal parts thereof and ranges from - 360° to 360°.

The angle is measured with the X-axis (G17 and G18) or the -Z-axis (G19).

Three possibilities are available:

1. B1= not programmed in the G77 block
In this case the sides of the pocket or groove are parallel to the main axes.
2. B1=0 in G77 block
In this case the axis of each pocket or groove is radial, thus lies in the direction of the radius from the centre of the circle to the point on the circle. Refer to example 3 for programming this case.

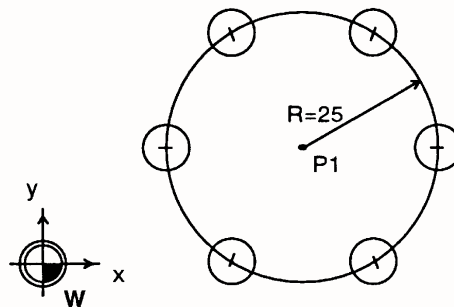
3. B1<>0 In G77 block

In this case B1= indicates the angle which the pocket or groove makes with the radius to the centre of the pocket. Refer to example 4 for programming this case.

Note: The word B1= has two meanings in a G77 block. Either it is the angle for rotating a pocket or groove or it is used to program the coordinates (B1=, L1= or X/Y with B1=) for the position of the centre of the circle.

Examples

EXAMPLE 1. Fixed cycle on a complete circle

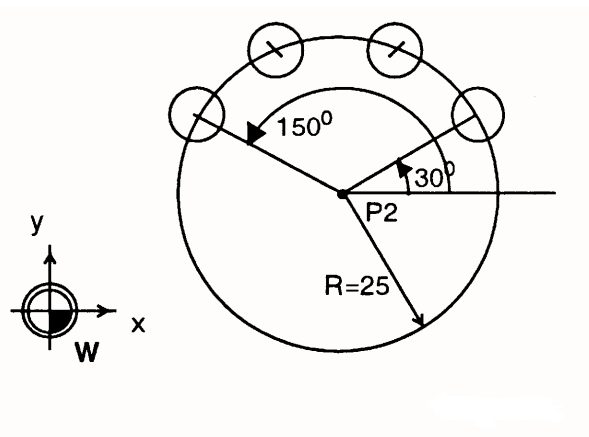


```
N30 G78 P1 X... Y... Z...
N40 T1 M6
N50 G81 Y1 Z-10 F100 S1000 M3
N60 G77 P1 R25 I0 J6
```

Explanation

N30: Definition of circle centre point (P1)
 N40: Load tool 1 (a drill)
 N50: Fixed cycle definition
 N60: Execute the fixed cycle on six points of the complete circle.

EXAMPLE 2. Fixed cycle on an arc



```

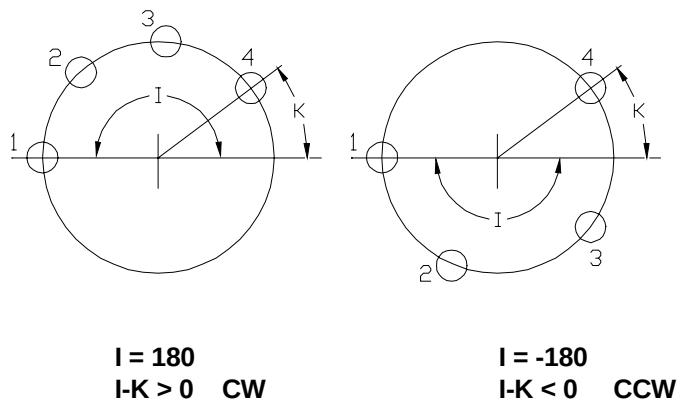
N30 G78 P2 X... Y... Z...
N40 T1 M6
N50 G81 Y1 Z-10 F100 S1000 M3
N60 G77 P2 R25 I30 K150 J4

```

Explanation

- N30: Definition of circle centre point (P2)
 N40: Load tool 1 (a drill)
 N50: Fixed cycle definition
 N60: Execute the fixed cycle on four equally spaced points on the circular arc, starting from 30

EXAMPLE 3. Direction holes on an arc



```

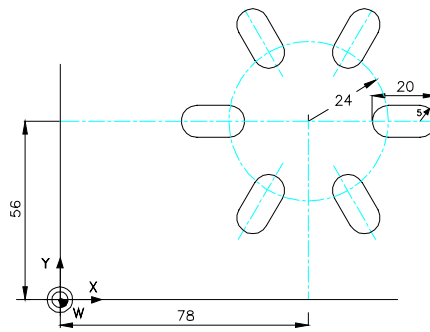
N50 G81 Y1 Z-10 F100 S1000 M3
N60 G77 X0 Y0 Z0 R25 I180 K30 J4
N70 G77 X0 Y0 Z0 R25 I-180 K30 J4

```

Explanation:

- N50: Cycle definition
 N60: Cycle repeating four times on an arc; Start on 180 degrees, End on 30 Grad in Clockwise (CW) direction.
 N70: Cycle repeating four times on an arc; Start on -180 degrees, End on 30 Grad in Counterclockwise (CCW) direction.

EXAMPLE 4. Radial grooves

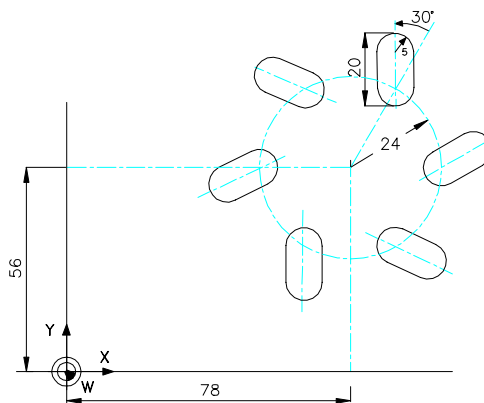


N60 T1 M6
 N65 G88 X20 Y10 Z-10 B1 F100 S1000 M3
 N70 G77 X78 Y56 Z0 R24 I0 J6 B1=0

Explanation

- N60: Load tool 1, a mill with a radius of 4.8 mm
 N65: Define the groove as if its sides are parallel to the X- and Y-axis
 N70: The radial grooves are milled. This block contains:
- the centre of the bolthole circle (X78, Y56, Z0),
 - the radius (R)
 - the angle, which the radius of the first point makes with the X-axis (I)
 - the number of holes on the circle (J)
 - B1=0 to indicate that the grooves are radial.

EXAMPLE 5. Rotated groove



N60 T1 M6
 N65 G88 X20 Y10 Z-10 B1 F100 S1000 M3
 N70 G77 X78 Y56 Z0 R24 I0 J6 B1=30

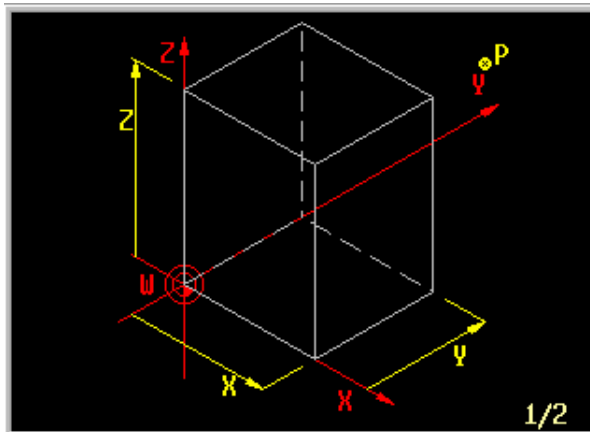
Explanation

- N60: Load tool 1, a mill with a radius of 4.8 mm
 N65: Define the groove as if its sides are parallel to the X- and Y-axis
 N70: The rotated grooves are milled.
 Refer to the previous example for an explanation of the addresses.
 Only the word B1= has a different meaning.

5.38 G78 Point definition

Purpose

Allows the coordinates of a point to be defined just once in a program. When a movement to the point is required, only the point number has to be programmed, not the point coordinates.



Format

N... G78 P... [Coordinates of point position]

Parameter

G	Point definition
X	Point coordinate
Y	Point coordinate
Z	Point coordinate
A	Point angle
B	Point angle
P	Point definition number
B2=	Polar angle
L2=	Polar length

Modal words

E...=

Associated functions

G0, G1, G2/G3, G45, G46, G77, G79, G92/G93, G145

Type of function

Non-modal

Notes and usage**COORDINATES**

Only Cartesian coordinates measured from the active program zero point W or polar coordinates (B2=, L2=..) in the main plane can be used.

G-FUNCTIONS WITH PRE-DEFINED POINTS

The following G-functions can only contain one defined point in their program block: G2/G3, G77, G92, G93.

The following G-functions can have a maximum of four defined points in their program block: G0, G1 and G79.

USING A PRE-DEFINED POINT

The format for using a predefined point is as follows:

N... G... P..., where P... represents the number of the point in the point memory.

Other formats are possible:

N... G79 P4=2 P2=10 P3=1 P1=5

N... G79 P1=E5 P2=E1

The P address can also be programmed with an index. The index value indicates the priority in the execution sequence. Index numbers 1 up to 4 are available (1=highest priority, 4=lowest priority). The number of the point in the point memory is entered behind the = sign. Another option would be parameterised entry of the point definition. In this event the index also indicates the priority.

POINT MEMORY

A maximum of 255 defined points can be stored in the CNC's Point Memory; a Machine Constant sets this maximum.

CANCELLATION

A defined point's coordinates remain active until:

- the point is redefined again by another G78-block;
- the point memory is changed or cleared by the user;
- a data carrier with defined points is read in.

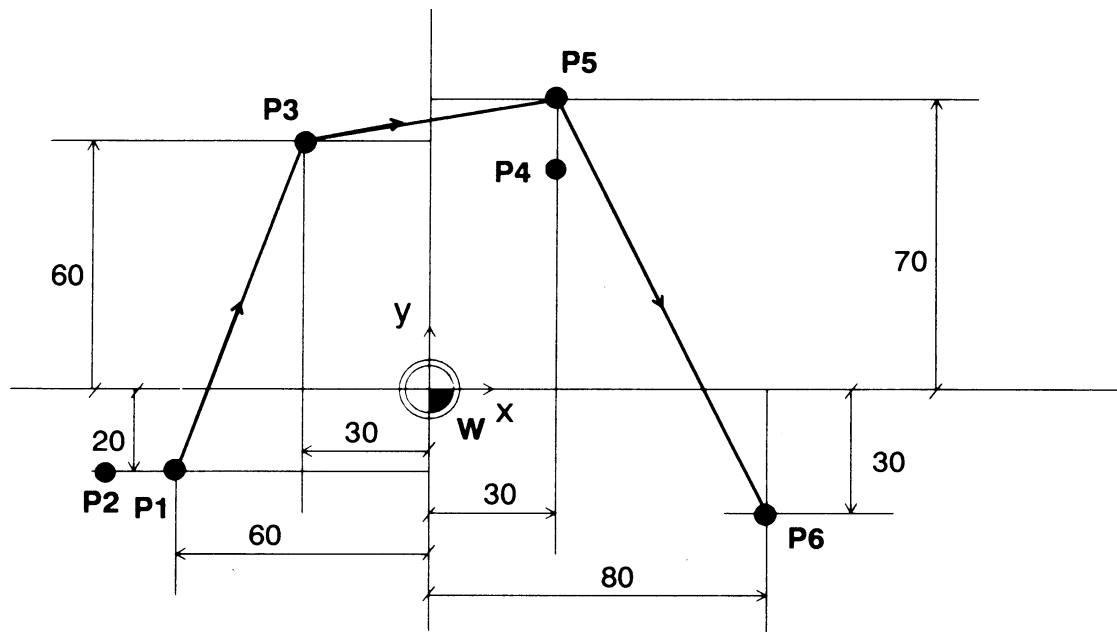
The Point Memory is not affected by CLEAR CONTROL.

RESTRICTIONS

Only one point can be specified in each G78-block, no other words are permitted.

Examples

EXAMPLE 1.



NB8548

```

...
N10 G78 P1 X=60 Y=-20
N11 G78 P2 X=70 Y=-20
N12 G78 P3 X=30 Y=60
N13 G78 P4 X=30 Y=50
N14 G78 P5 X=30 Y=70
N15 G78 P6 X=80 Y=-30
:
N90 G0 P1=1
N91 G1 P1=3 P2=5 P3=6 F1000

```

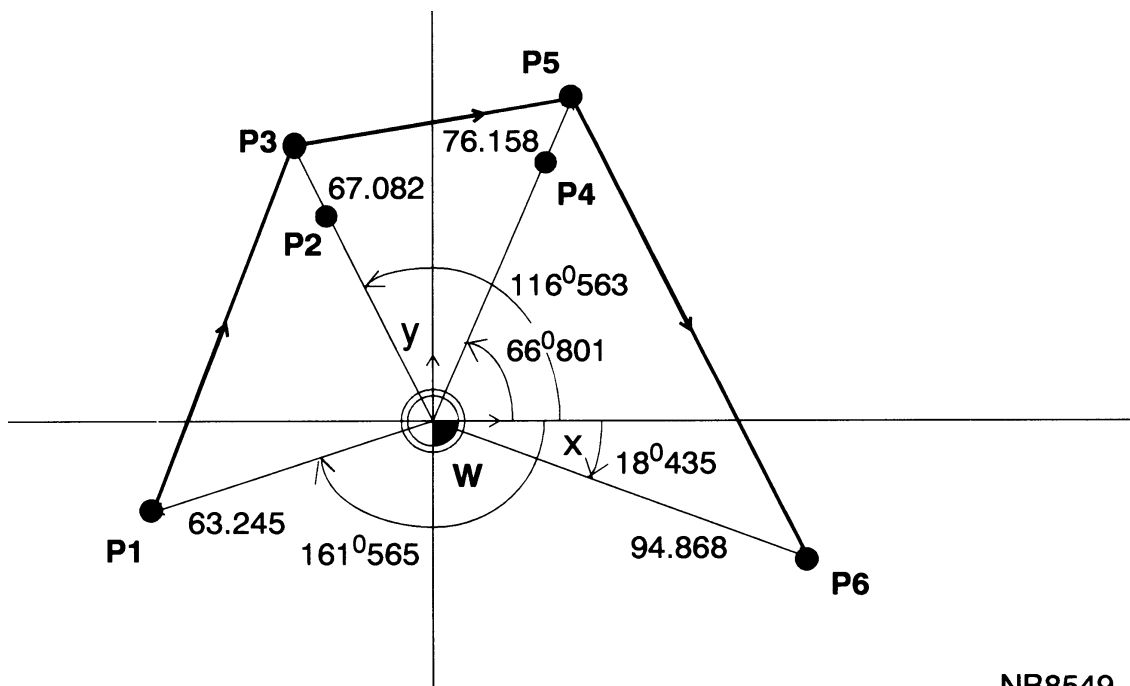
Explanation:

N10-N15: The points are defined.

N90: Move tool rapidly to position defined by P1.

N91: Move tool at set feedrate, first to P3, then to P5 and finally to P6.

EXAMPLE 2. Using polar coordinates



NB8549

```

:
N10 G78 P1 B2= -161.565 L2= 63.245
N11 G78 P2 B2= 116.563 L2= 65
N12 G78 P3 B2= 116.563 L2= 67.082
N13 G78 P4 B2= 66.801 L2= 72
N14 G78 P5 B2= 66.801 L2= 76.158
N15 G78 P6 B2= - 18.435 L2= 94.868
:
N90 G0 P1=1
N91 G1 P1=3 P2=5 P3=6 F1000

```

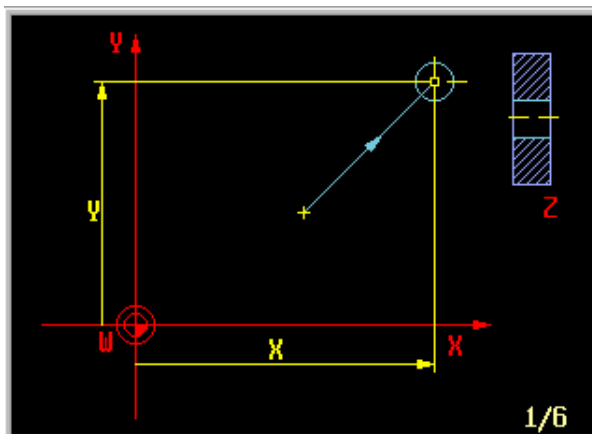
Explanation:

N10-N15: The points are defined.
 N90: Move tool rapidly to position defined by P1.
 N91: Move tool at set feedrate, first to P3, then to P5 and finally to P6.

5.39 G79 Activate cycle

Purpose

To activate any fixed cycle which controls hole operations (G81, G83 to G86) or milling operations (G87 to G89) at programmed positions.



Format

N... G79 [Axis coordinates] {B1=...}

Parameters

G Activate cycle
 X Point coordinate
 Y Point coordinate
 Z Point coordinate
 A Point angle
 B Point angle
 B1= Angle
 B2= Polar angle
 ?90= Point abs. (X,Y,Z..)
 ?91= Point incr. (X,Y,Z..)
 L1= Path length
 L2= Polar length
 P1= Point definition number
 P2= Point definition number

P3= Point definition number
 P4= Point definition number

Modal Parameters

F, S, T, M, H, E...=

T1=, T2=

Associated functions

G77, G81, G83-G89

Type of function

Non-modal.

Notes and usage

EXECUTING DEFINED FIXED CYCLES

The positions where a previously defined fixed cycle is to be executed are programmed in the G79-blocks, which follow the fixed cycle definition.

A fixed cycle for a hole operation (G81, G83-G86) is executed in the tool axis which is perpendicular to the main plane defined by the G- function for plane selection (G17, G18 or G19).

A fixed milling cycle is executed in the main plane defined by the G- function for plane selection (G17, G18 or G19).

The first G79-block, which follows a defined fixed cycle, must contain a tool axis coordinate.

The direction of the depth operation is programmed in the fixed cycle block with the sign of the Z-word indicating the total depth.

POSITIONING LOGIC

To reduce the possibility of collision between tool and workpiece the positioning logic is available. Refer to G0 POSITIONING LOGIC for additional information.

In addition, the programmer must ensure that the tool cannot collide with any device holding the workpiece in position.

SPINDLE ROTATION

If the spindle is not rotating when a G79-block is activated, an error message is generated and program execution stopped.

RADIUS COMPENSATION

When a G79-block is activated, radius compensation is cancelled by the CNC automatically generating the G40-function.

If radius compensation is required after a G79-block, the appropriate function (G41 -G44) must be programmed.

ROTATED POCKET OR GROOVE (B1=)

A previously defined pocket or groove (G87 or G88) can be rotated about an angle. The centre of rotation is the point used in the G79 block to program the location of the pocket or groove.

The angle is programmed with the word B1= in degrees and decimal parts thereof and ranges from - 360 to 360'.

The angle is measured with the X-axis (G17 and G18) or the -Z-axis (G19).

If B1=0 is programmed or B1= is not programmed at all, the pocket or groove are milled axis parallel.

Note: The word B1= has two meanings in a G79 block. Either it is the angle for rotating a pocket or groove or it is used to program the coordinates (B1=, L1= or X/Y, B1=) for the position of the centre of the pocket.

GROUP FUNCTION (G0, G1, G2, G3) AND G6

A G79-block ignores the functions G0, G1, G2, G3 and G6. When the G79-block execution is finished, the concerning function will become active again.

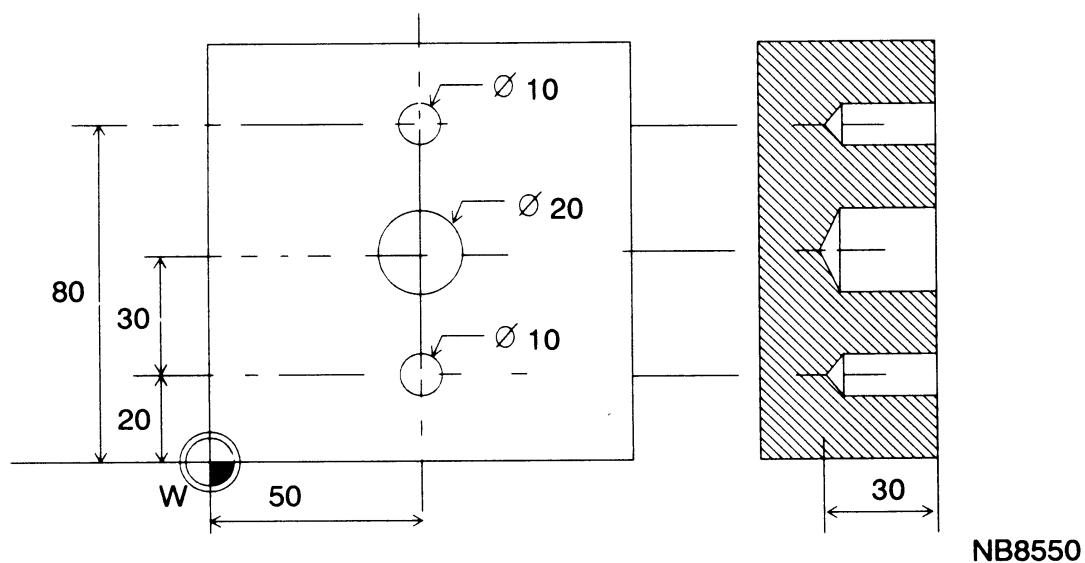
TOOL POINTING IN POSITIVE DIRECTION (G66/G67)

If the tool is pointing in the positive direction of the tool axis (G67 activated), the depth of the fixed cycle must be programmed with a positive sign (+) to indicate that the cycle is to be executed in the positive direction of the tool axis.

With the milling cycles the direction of rotation on the circular arcs is automatically changed in the opposite direction.

The milling direction programmed with the J-word, is not automatically changed.

Examples



EXAMPLE 1. Three holes to be drilled

```

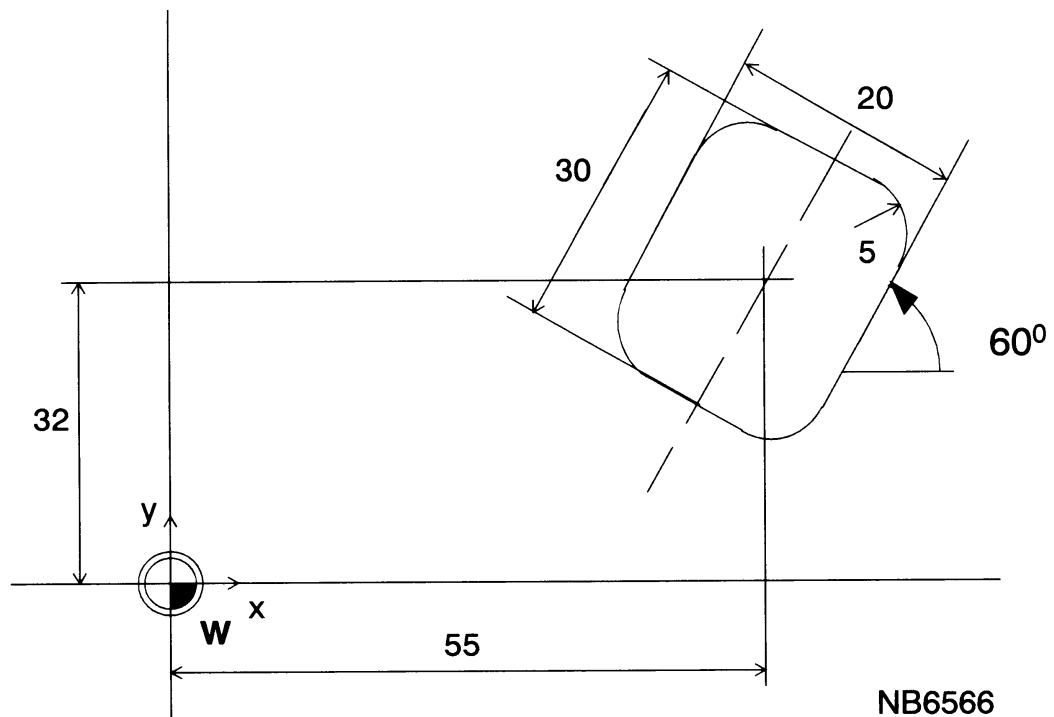
N50 G78 P1 X50 Y20 Z0
N55 G78 P2 X50 Y80 Z0
N60 T1 M6
N65 G81 Y1 Z-30 F100 S1000 M3
N70 G79 P1=1 P2=2
N75 T2 M6
N80 G79 X50 Y50 Z0
N90 M30

```

Explanation

N50: Define point 1
 N55: Define point 2
 N60: Load tool 1 (drill of diameter 10)
 N65: Define drilling cycle and start the spindle
 N70: Drill holes at points 1 and 2
 N75: Load tool 2 (drill of diameter 20)
 N80: Drill the hole
 N90: End of program.

EXAMPLE 2. A rotated pocket



```

:
N55 G17
N60 T1 M6
N65 G87 X30 Y20 Z-5 B1 R5 F100 S1000 M3
N70 G79 X55 Y32 Z0 B1=60
:

```

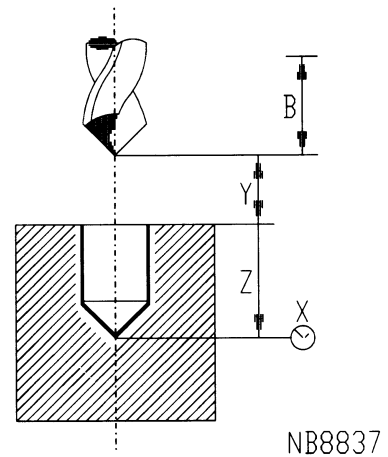
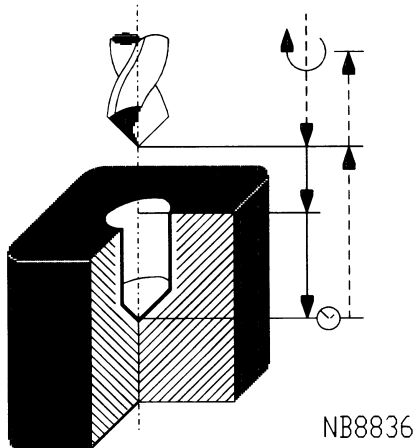
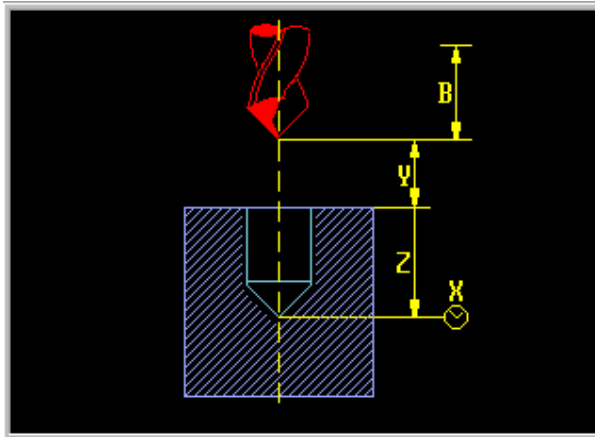
Explanation

N55: Define the main plane for the pocket
 N60: Load tool 1, a mill with a radius of 4.5 mm
 N65: Define the pocket as if its sides are parallel to the X- and Y-axis
 N70: The pocket is milled. This block contains the centre of the pocket (X55, Y32, Z0) and the angle (60°) the axis of the pocket makes with the X-axis.

5.40 G81 Drilling cycle

Purpose

To define in one program block the drilling of a hole.



Format

N... G81 Z... {Y...} {X...} {B...}

Parameters

G	Drilling cycle
X	Dwell time (sec)
Y	Clearance
Z	Drilling depth
B	Retract distance

Modal parameters
F, S, T, M, H, E...=
T1=, T2=

Type of function

Modal

Associated functions

G77, G79, G83-G89

Notes and usage

DEPTH OF OPERATION (Z)

Final depth of operation measured from the surface.

The sign of the Z-word indicates the direction of depth movement in the tool axis:

"-" in the negative direction, in most cases into the hole

"+" in the positive direction.

RETRACT DISTANCE (B)

The retract distance is added to the clearance value (Y-word). It can be used e.g. to avoid obstacles. This extra distance can have either a positive or negative value.

If the B-word is not programmed, the retract movement is executed to a point the clearance distance above the surface.

DWELL AT BOTTOM OF HOLE (X)

If required, a dwell at the bottom of the hole can be programmed in steps of .1 second.

Minimum programmable dwell period: .1 second

Maximum programmable dwell period: 900 seconds

If the X-word is not programmed, no dwell is executed.

EXECUTION

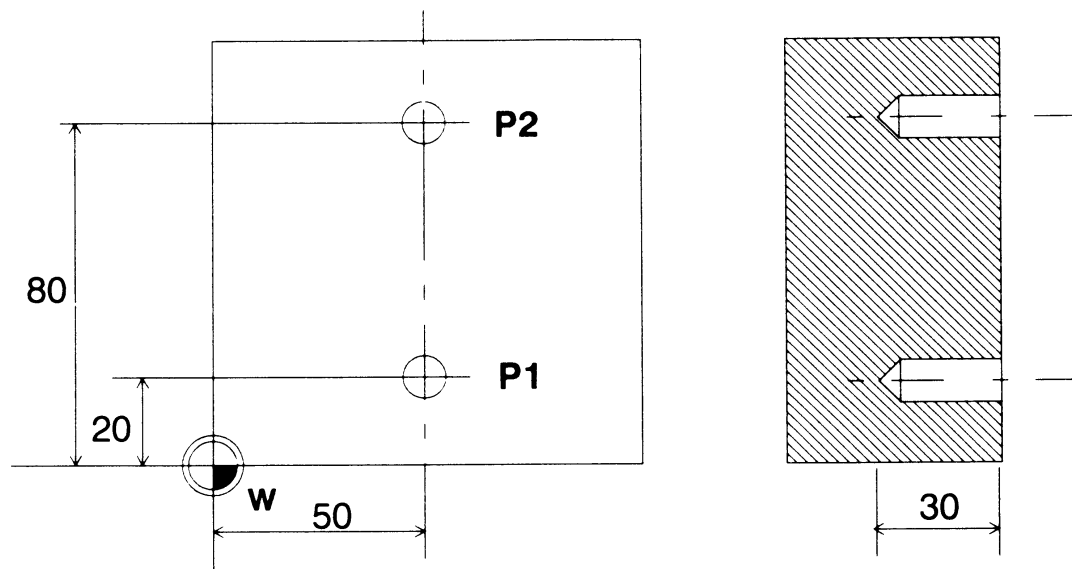
A defined fixed cycle is executed on the position(s) programmed in either the G77 or G79 block.

The cycle is executed in the tool axis, which is stated with the function for plane selection (G17, G18 or G19).

CANCELLATION

The cycle's values are cancelled when a new cycle is defined or by Softkey CLEAR CONTROL, M30 or by Softkey CANCEL PROGRAM.

Example



NB8552

```

N50 G78 P1 X50 Y20 Z0
N55 G78 P2 X50 Y80 Z0
N60 G0 Z10 T1 M6
N65 G81 X1.5 Y1 Z-30 F100 S500 M3
N70 G79 P1 P2

```

Explanation

N50: Define point 1
 N55: Define point 2
 N60: Load tool 1 and move tool out the tool change position
 N65: Define fixed drilling cycle and start the spindle
 N70: Execute fixed cycle at point 1 and then point 2.

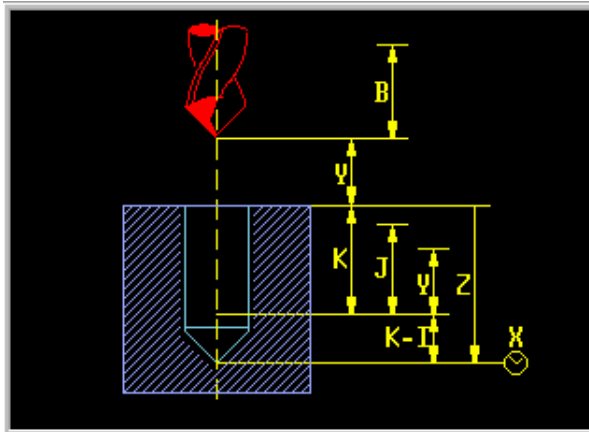
Fixed cycle sequence:

- drill moves at rapid traverse rate to a point the clearance distance (Y-word of G81) above the surface.
- drill feeds to depth (Z-word of G81) at set feedrate
- drill is stopped for 1.5 seconds
- drill is retracted at rapid traverse rate to a point the clearance distance above the surface.

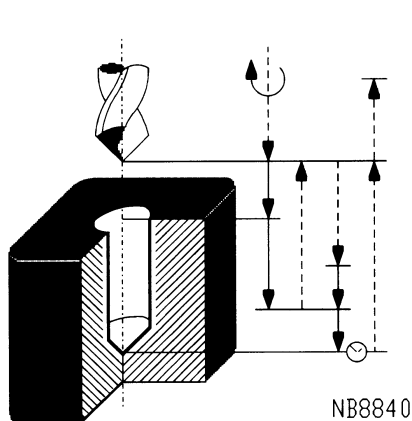
5.41 G83 Deep hole drilling cycle

Purpose

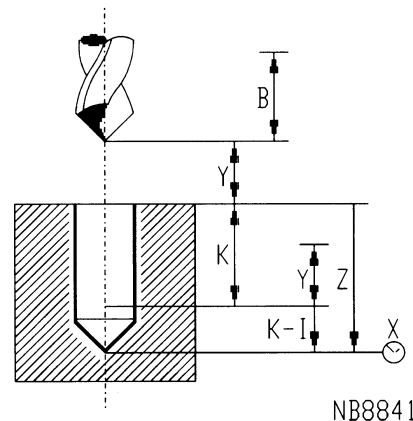
To define in one program block the drilling of a deep hole.



Format



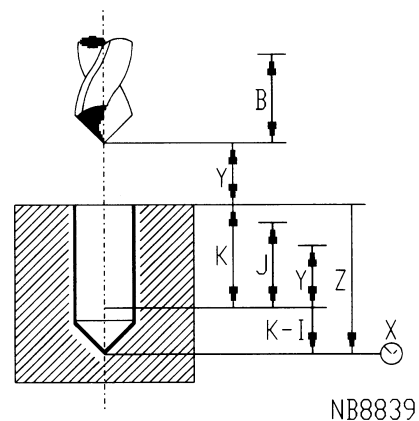
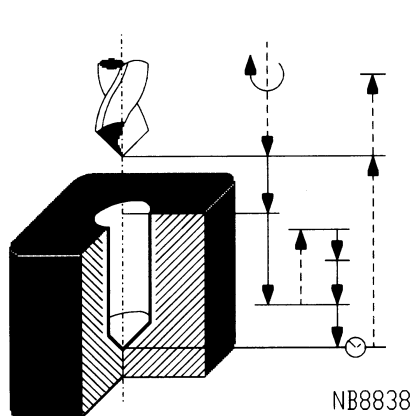
NB8840



NB8841

Tool movements when J0 is active (tool is fully retracted to clearance position Y)

N... G83 Y... Z... K... J0 {I...} {X...} {B...}



Tool movements when $J > 0$ (tool remains inside workpiece between cutting passes)

N... G83 Y... Z... K... J... {I...} {X...} {B...} {K1...}

Parameters

G	Deep hole drilling cycle
X	Dwell time (sec)
Y	Clearance
Z	Overall drilling depth
B	Retract distance
I	Drilling depth decrement
J	Retract distance after step
K	Drilling depth first movement
K1=	Number of retract distances

Modal parameters

F, S, T, M, H, E...=

T1=, T2=

Associated functions

G77, G79, G81, G84-G89

Type of function

Modal.

Notes and usage

EXECUTION

A defined fixed cycle is executed on the position(s) programmed in either the G77 or G79 block.

The cycle is executed in the tool axis, which is stated with the function for plane selection (G17, G18 or G19).

DEPTH OF OPERATION (Z)

Final depth of operation measured from the surface. The sign of the Z-word indicates the direction of depth movement in the tool axis:

"-" in the negative direction, in most cases into the hole

"+" in the positive direction.

FIRST FEED-IN DISTANCE (K)

In general a deep hole drilling operation takes place in several steps. The drilling depth of the first step is programmed with the K-word. If the K-value is greater than the total depth (Z-word), the hole is drilled at depth in one cutting pass.

The K-word has no sign.

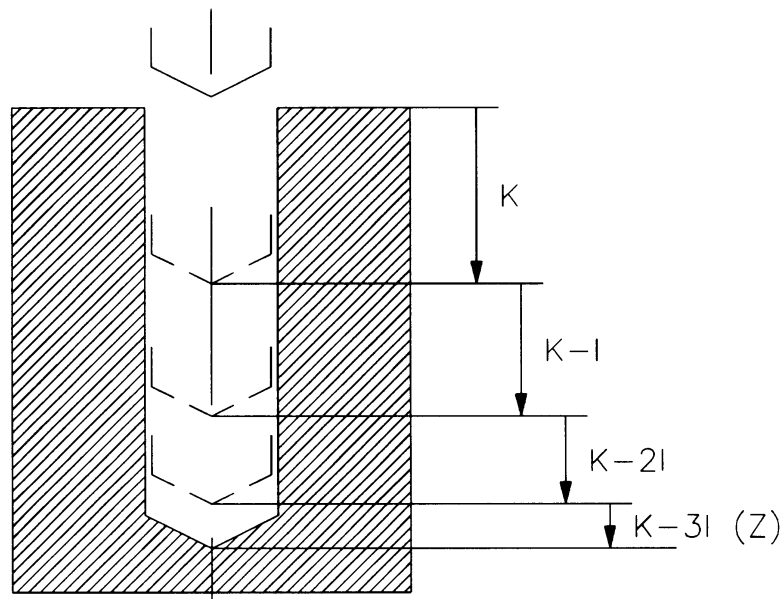
DWELL AT BOTTOM OF HOLE (X)

If required, a dwell at the bottom of the hole can be programmed in steps of .1 second.

Minimum programmable dwell period: .1 second

Maximum programmable dwell period: 900 seconds

If the X-word is not programmed, no dwell is executed.



NB8854a

INCREMENTAL REDUCTION (I)

If the calculated feed-in distance becomes less than the I-value, the I-value is used instead. The final feed-in distance can be smaller than the I-value.

The I-word has no sign.

I0: all feed-in distances (except perhaps the final one) are equal to the first feed-in distance (K-word).

RETRACT DISTANCE (B)

The retract distance is added to the clearance value (Y-word). It can be used e.g. to avoid obstacles. This extra distance can have either a positive or negative value.

If the B-word is not programmed, the retract movement is executed to a point the clearance distance above the surface.

RETRACT INDICATOR (J)

A special word (J) is used to indicate how the tool is to be retracted after each cutting pass:

J0: after each cutting step, the tool is retracted out of the hole to the point defined by the clearance distance.

J>0: after each cutting step, the tool is retracted over the programmed distance. In this way chips are broken, but the tool remains in the hole.

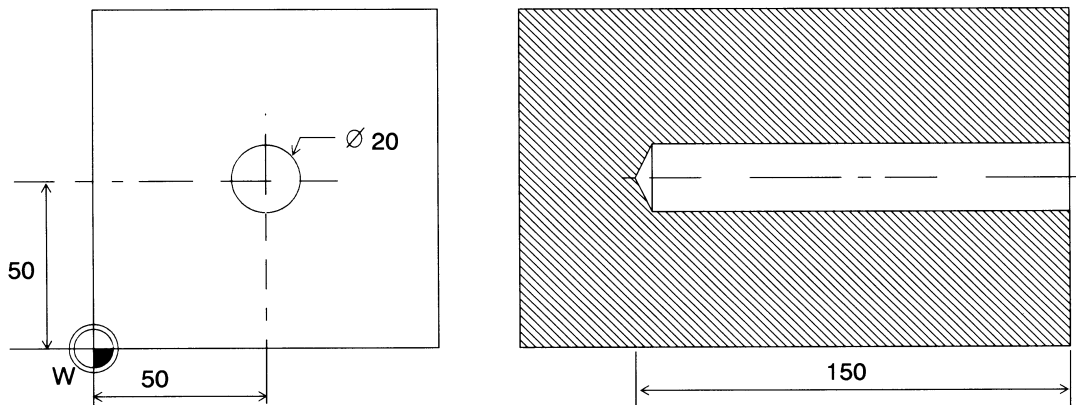
The J-word has no sign.

NUMBER OF SPECIAL RETRACT DISTANCE (K1=)

After a programmable number of feed movements defined by (K1 =), a retract will take place to a position the clearance distance before the preceding depth, to remove the chips. This only will take place if the special retract distance (J>0) is defined.

CANCELLATION

The cycles values are cancelled when a new cycle is defined or by Softkey CLEAR CONTROL, M30 or by Softkey CANCEL PROGRAM.

Example

NB8553

EXAMPLE 1:

```

:
N5 T1 M6
N10 G83 Y4 Z-150 I2 J6 K20 F200 S500 M3
N20 G79 X50 Y50 Z0

```

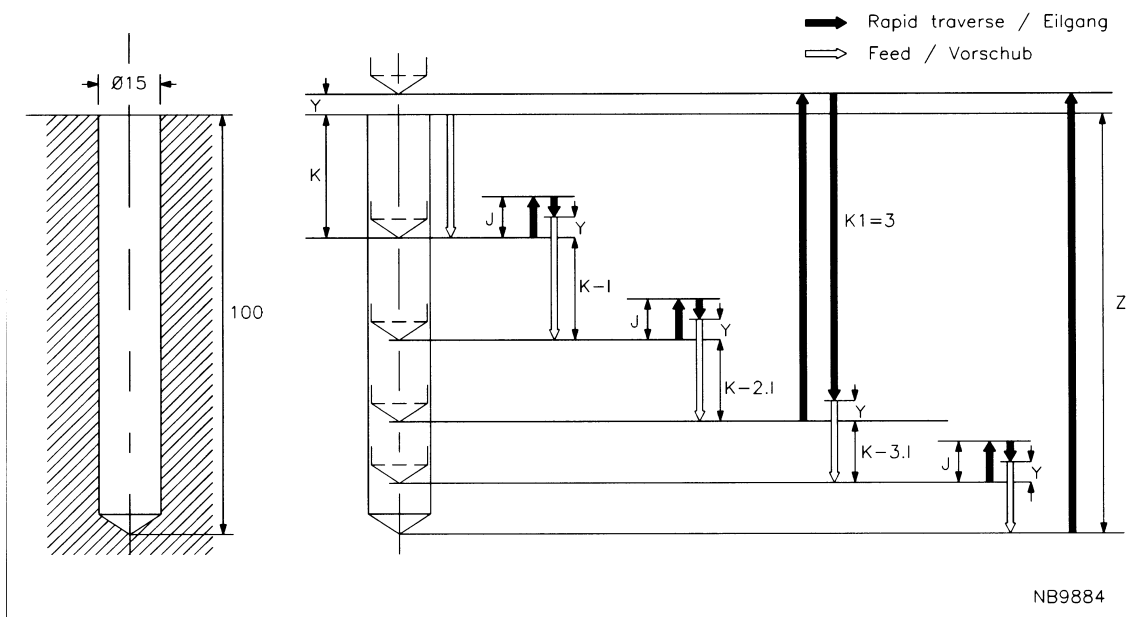
N5: Load tool 1.

N10: Define the fixed cycle for deep hole drilling.

N20: Execute the fixed cycle at the programmed position

EXAMPLE 2:

Make a deep hole drilling (d=15, depth 100). After 3 feed movements a retract must take place.



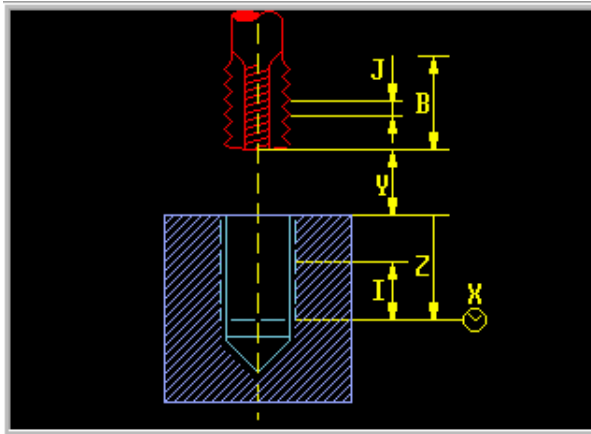
:
N.. G83 Y3 Z-100 I5 J6 K30 K1=3

G83 Deep hole drilling cycle
Y3 Clearance distance 3 mm
Z-100 Depth of operation 100 mm
I5 Incremental reduction 5 mm
K30 First feed-in distance 30 mm. K-Value is reduced with the I-Value every cutting pass.
K1=3 After 3 feed movements a retract will take place to the clearance position.

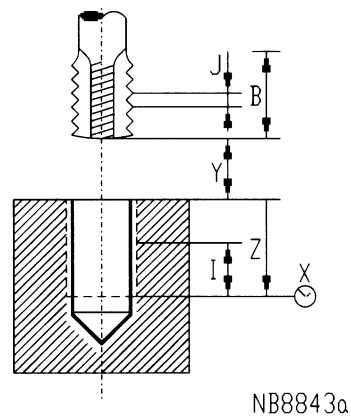
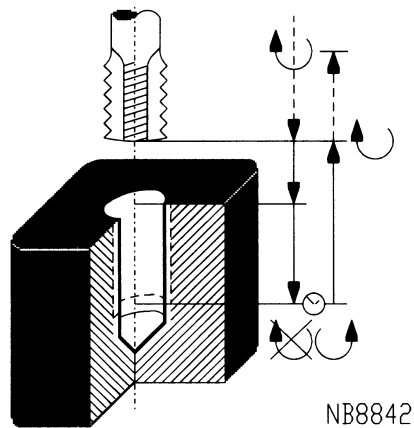
5.42 G84 Tapping cycle

Purpose

To define in one program block the tapping of a hole.



Format



N... G84 Z... {Y...} {B...} {J...} {X...}

or

N... G84 I1=0 Z... {Y...} {B...} {J...} {X...}

The tapping cycle can also be done as an interpolation in a close control circuit between the tool axis and the spindle. In this interpolation the acceleration power of the spindle is carrying out. There by is guaranteed, that the spindle is running with the desired speed. ("Rigid tapping").

Parameters

```

G    Tapping cycle
X    Dwell time (sec)
Y    Clearance
Z    Tapping depth
B    Retract distance
I    Speed ramp (rev)
J    Pitch
I1=  Interpolation (0=without,1=with)

```

I1= Interpolation (0 = without interpolation, 1 = with interpolation)

Modal parameters

F, S, T, M, H, E...= , T1=, T2=,

Type of function

Modal

Associated functions

G77, G79, G81, G83, G85-G89

Notes and usage**INTERPOLATION (I1=)**

Tapping can be done without or with interpolation.

I1=0 without interpolation (default, open position control circuit)

I1=1 with interpolation (close position control circuit)

An active "tilting working plane (G7)" can work only with interpolation (I1=1).

It is also possible, by an active "tilting working plane (G7)" and when the tool head is not tilted (Tool axis is equal to Z-axis), to work without interpolation (I1=0).

TAPPING DEPTH (Z)

Final tapping depth measured from the surface. The sign of the Z-word indicates the direction of the movement in the tool axis:

FLOATING TAP HOLDER

When a floating tap holder is used, the clearance distance (Y) must be sufficient for the tool not to touch the workpiece when the tool is fully retracted and the tap holder spring is no longer under compression.

RETRACT DISTANCE (B)

The retract distance is added to the clearance value (Y-word). It can be used e.g. to avoid obstacles. This extra distance can have either a positive or negative value.

If the B-word is not programmed, the retract movement is executed to a point the clearance distance above the surface.

RIGID TAPPING

If a transducer is mounted at the spindle of the machine tool, rigid tapping, thus tapping without using a floating tapholder, is possible.

To eliminate the drift of the spindle an oriented spindle stop (M19) must be programmed before the tapping starts. On some machine tools the oriented spindle stop is automatically executed with a tool change (M6). Refer to the machine tool builder's documentation for details.

RETAP THREAD

On machines with interpolation gives the programming of an oriented spindle stop (M19) the possibility to retap the thread.

Note:

After the interpolierende retap thread (I1=1) the modal M-function (M3, M4) is not active. This M function will be overwritten by M19.

SPEED RAMP (REV.) (I)

This word is used by the CNC to determine the point where the spindle starts to safely slow down before the end of the thread is reached and stop at the bottom of the thread.

With the I-address the number of revolutions required for the spindle to safely slow down and stop at the bottom of the thread is programmed. If the I-word is not programmed, the CNC uses a Machine Constant value (MC723) for establishing this point.

PITCH OF THE THREAD (J)

The pitch of the thread can be programmed:

- by using the J-word
- by programming the F (feedrate) = pitch (J) * spindle speed (S)

DWELL AT BOTTOM OF HOLE (X)

If required, a dwell at the bottom of the hole can be programmed in steps of 0.1 seconds.

Minimum programmable dwell period: .1 second

Maximum programmable dwell period: 900 seconds

If the X-word is not programmed, no dwell is executed.

MINIMUM SPINDLE SPEED

When feed and speed during G84 will be reducing to zero, it is possible that the spindle do not rotate. To avoid this is a minimum spindle speed in a machine constant (MC727) defined.

EXECUTION

A defined fixed cycle is executed on the position(s) programmed in either the G77 or G79 block. The cycle is executed in the tool axis, which is stated with the function for plane selection (G17, G18 or G19).

CANCELLATION

The cycle's values are cancelled when a new cycle is defined or by Softkey CLEAR CONTROL, M30 or by Softkey CANCEL PROGRAM.

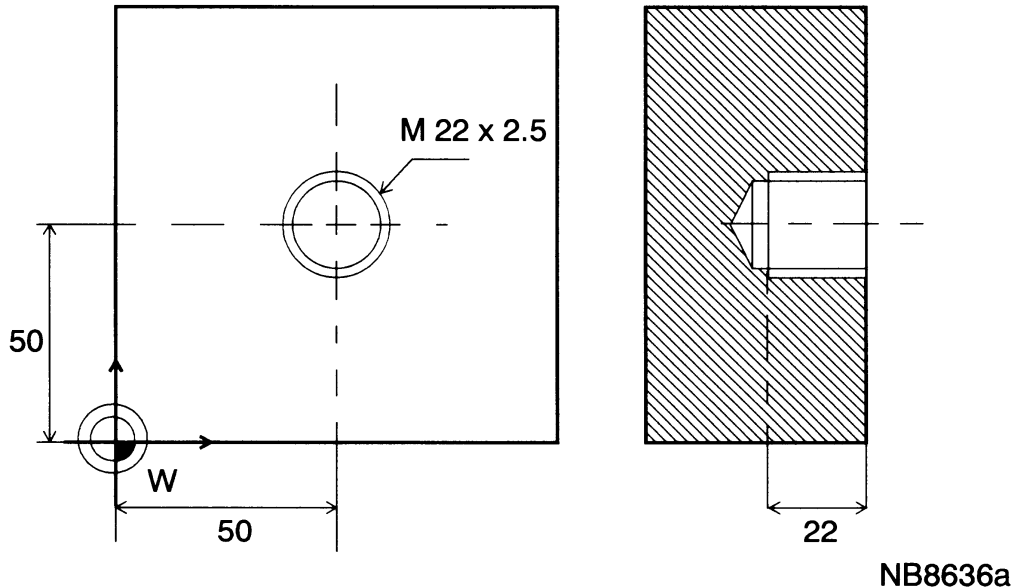
Note: In case of G84 execution by G79 the control must be in G94 mode (F in mm/min) and not in G95 mode (F in mm/rev). The programmer should also always program G94 before G84.

MACHINE CONSTANT

The machine constant (MC723 and MC727) will not be use by interpolating.

The machine constant for the spindle must be defined properly. The acceleration of the spindle is calculated by machine constant MC2491, 2521,2551,2581 and MC2495, 2525, 2555, 2585. For an optimal control must be active also MC4430.

Example



```
N14 T3 M6
N15 G84 Y9 Z-22 J2.5 S56 M3 F140
N20 G79 X50 Y50 Z0
```

Explanation:

N14: Load tool 3 (Tap M22 x 2.5).
 N15: Define tapping cycle and start the spindle.
 N20: Execute the tapping cycle on the programmed position. A floating tap holder is used.

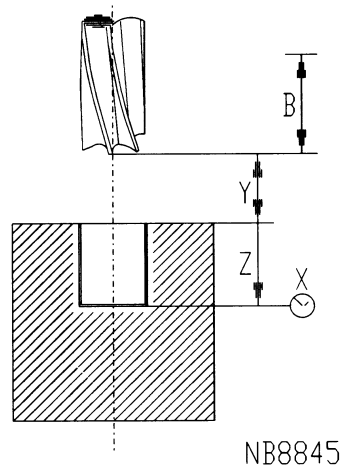
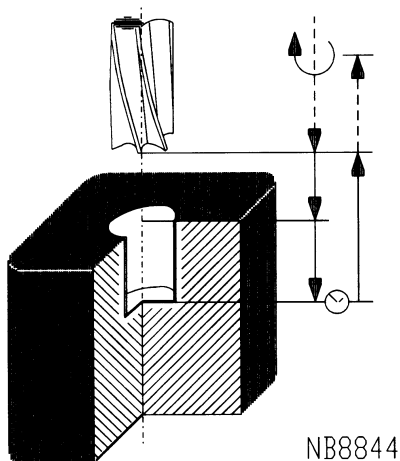
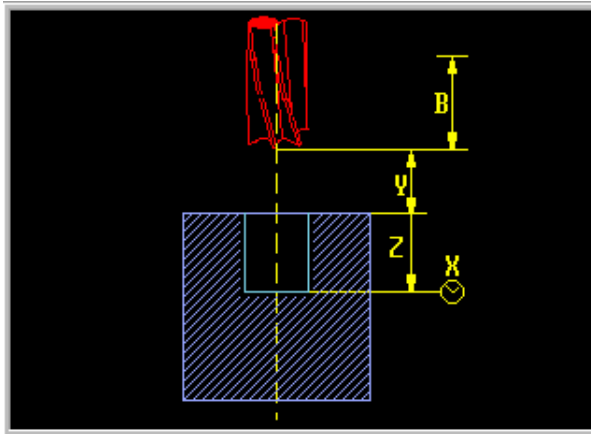
Tapping cycle sequence:

- tap moves at rapid traverse rate to a point the clearance distance (Y-Word) above the surface. Spindle rotation clockwise (M3).
- tap feeds to tapping depth at a set feedrate. (The feed depends on pitch and spindle speed).
- retract the tap to a point the clearance distance above the surface after the spindle rotation is switched to counter-clockwise.

5.43 G85 Reaming cycle

Purpose

To define in one program block the reaming of a hole.



Format

N... G85 Z... {Y...} {B...} {X...} {F2=..}

Parameters

G	Reaming cycle
X	Dwell time (sec)
Y	Clearance
Z	Reaming depth
B	Retract distance
F2=	Feedrate to startpoint

Modal parameters
F, S, T, M, H, E...=
T1=, T2=

Type of function

Modal

Associated functions

G77, G79, G81, G83, G84, G86-G89

Notes and usage

REAMING DEPTH (Z)

Reaming depth measured from the surface.

The sign of the Z-word indicates the direction of depth movement in the tool axis:

"-" in the negative direction, in most cases into the hole

"+" in the positive direction.

RETRACT DISTANCE (B)

The retract distance is added to the clearance value (Y-word). It can be used e.g. to avoid obstacles. This extra distance can have either a positive or negative value.

If the B-word is not programmed, the retract movement is executed to

FEEDRATE TO STARTPOINT (F2=)

Feedrate to startpoint (F2=) is the feed from depth to starting position. This Feed (F2=) allows a faster retract feed and can be programmed. This will reduce the overall cycle execution duration. If the Feed F2= is not defined in a cycle the Feed to the starting position, is the programmed Feed (F).

DWELL AT BOTTOM OF HOLE (X)

If required, a dwell at the bottom of the hole can be programmed in steps of .1 second.

Minimum programmable dwell period: .1 second

Maximum programmable dwell period: 900 seconds

If the X-word is not programmed, no dwell is executed.

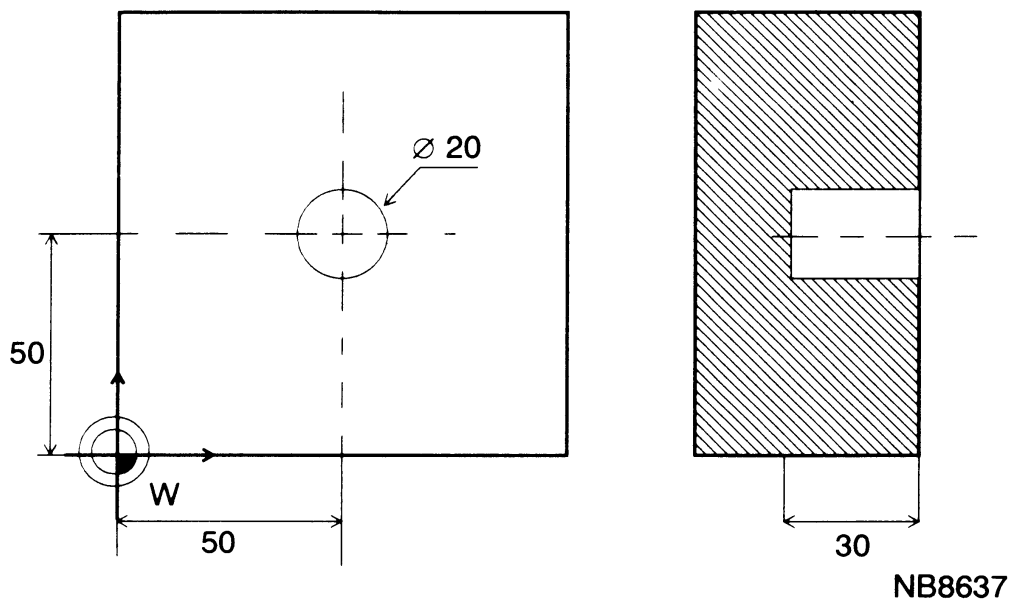
EXECUTION

A defined fixed cycle is executed on the position(s) programmed in either the G77 or G79 block. The cycle is executed in the tool axis, which is stated with the function for plane selection (G17, G18 or G19).

CANCELLATION

The cycles values are cancelled when a new cycle is defined or by Softkey CLEAR CONTROL, M30 or by Softkey CANCEL PROGRAM.

Example



```

N25 Z10 T4 M6
N30 G85 X2 Y3 Z-20 F100 S1000 F2=200 M3
N35 G79 X50 Y50 Z0

```

Explanation:

N25: Load tool 4, the reamer
 N30: Define the reaming cycle and start the spindle
 N35: Execute the reaming cycle at the programmed position

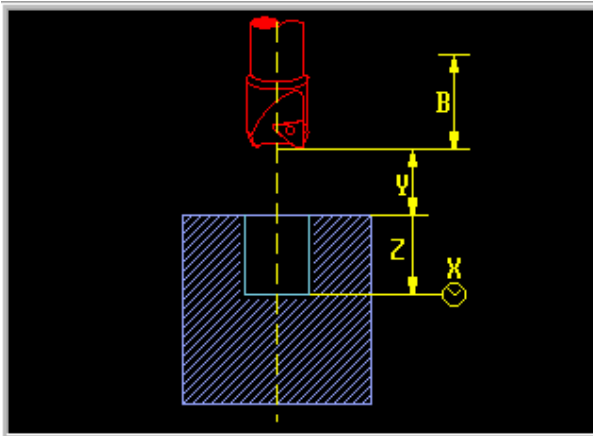
Cycle sequence:

- the reamer moves at rapid traverse rate to the clearance distance (Y3)
- the reamer feeds to depth (Z-20) at feedrate (F50)
- a dwell of 2 seconds (X2) at the bottom of the hole
- the reamer is retracted at feedrate (F2=200) to the clearance distance (Y3)

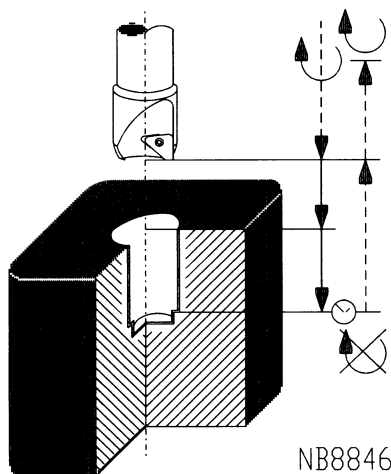
5.44 G86 Boring cycle

Purpose

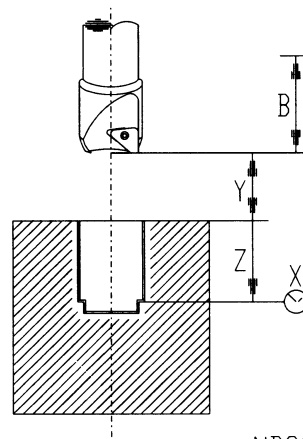
To define in one program block the boring of a hole.



Format



NB8846



NB8847

N... G86 Z... {Y...} {X...} {B...}

Parameters

G	Boring cycle
X	Dwell time (sec)
Y	Clearance
Z	Boring depth
B	Retract distance

Modal parameters

F, S, T, M, H, E...=

T1=, T2=

Type of function

Modal

Associated functions

G77, G79, G81, G83-G85, G87-G89

Notes and usage

BORING DEPTH (Z)

Boring depth measured from the surface.

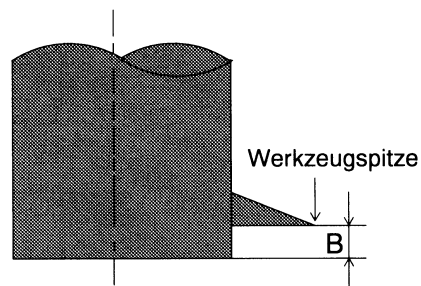
The sign of the Z-word indicates the direction of depth movement in the tool axis:

"-" in the negative direction, in most cases into the hole

"+" in the positive direction.

RETRACT DISTANCE (B)

The retract distance is added to the clearance value (Y-word). It can be used to compensate for a bore whose tool tip is not on the bottom of the boring bar.



NB8551

If this extra distance is not added, the bottom of the boring bar could still be inside the workpiece after the tool tip has been retracted the clearance distance.

If the B-word is not programmed, the tool tip is retracted to the clearance distance above the surface.

DWELL AT BOTTOM OF HOLE (X)

If required, a dwell at the bottom of the hole can be programmed in steps of .1 second.

Minimum programmable dwell period: .1 second

Maximum programmable dwell period: 900 seconds

If the X-word is not programmed, no dwell is executed.

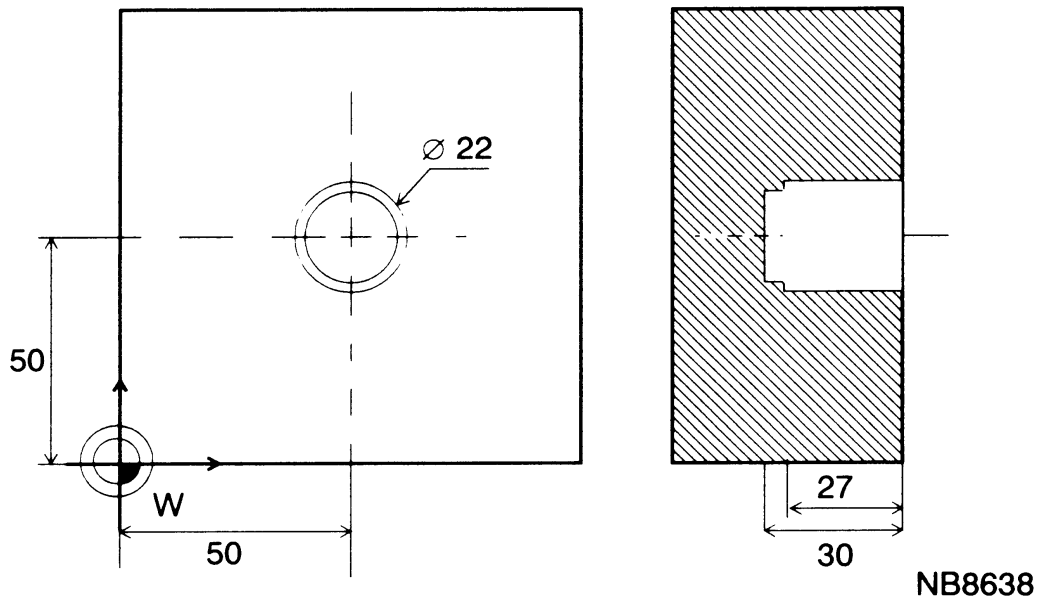
EXECUTION

A defined fixed cycle is executed on the position(s) programmed in either the G77 or G79 block.

The cycle is executed in the tool axis, which is stated with the function for plane selection (G17, G18 or G19).

CANCELLATION

The cycles values are cancelled when a new cycle is defined or by softkey CLEAR CONTROL, M30 or by softkey CANCEL PROGRAM.

Example

```
N45 T5 M6
N50 G86 X1 Y9 Z-27 B10 F100 S500 M3
N55 G79 X50 Y50 Z0
```

Explanation:

N45: Load tool 5, a boring bar.
 N50: Define boring cycle and start the spindle.
 N55: Execute fixed cycle at the programmed position

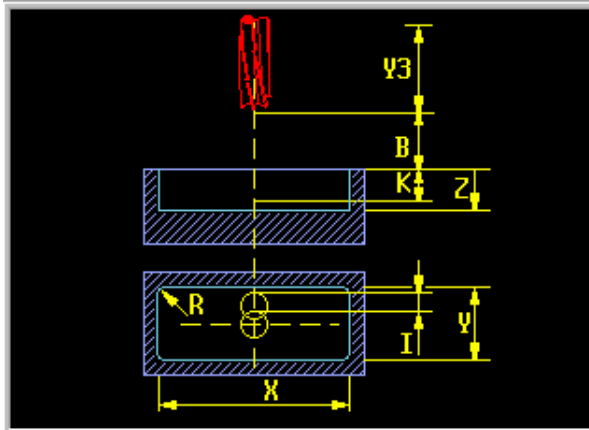
Fixed cycle sequence:

- the tool tip of the bore on the boring bar moves at rapid traverse rate to the clearance distance (Y-word).
- bore feeds to depth (Z-word) at feedrate
- at depth the spindle is stopped
- a dwell of 1 second
- the tool tip of the bore is retracted at rapid traverse rate to the clearance distance (Y-word).
- The tool tip is out of the hole.
- the spindle is started again
- retract the tool tip to a point the retract distance (B-word) above the surface.

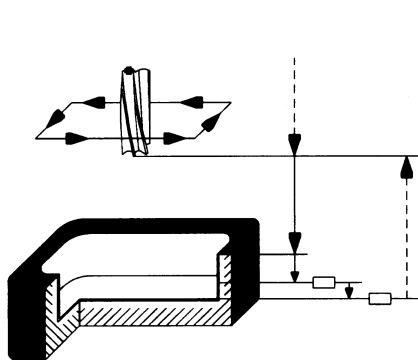
5.45 G87 Rectangular pocket milling

Purpose

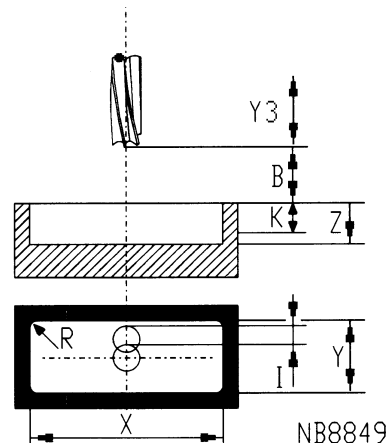
To define in one program block the geometry of a rectangular pocket and some parameters for cutting the pocket.



Format



NB8848



NB8849

N... G87 X... Y... Z... {B...} {R...} {J...} {I...} {K...} {Y3=..} {F2=..}

Parameters

G	Rectangular pocket milling cycle
X	Dimension parallel to X
Y	Dimension parallel to Y
Z	Total pocket depth
B	Clearance
I	Cutting width mill in %
J	J1:climb / J-1:conventional
K	Cutting depth
R	Corner radius
Y3=	Special retract distance
F2=	In depth feed (only this block)

Pocket geometry

X	Dimension parallel to X
Y	Dimension parallel to Y
Z	Total pocket depth
R	Corner radius
F2=	In depth feed (only this block)

Cutting parameters

B	Clearance
I	Cutting width mill in %
J	J1: climbing / J-1: conventional
K	Cutting depth
Y3=	Special retract distance

Modal parameters

F, S, T, M, H, E...=
T1=, T2=

Type of function

Modal

Associated functions

G77, G79, G81, G83-G86, G88, G89

Notes and usage**TOTAL POCKET DEPTH (Z)**

Total pocket depth measured from the surface.

The sign of the Z-word indicates the direction of depth movement in the tool axis:

"_" in the negative direction, in most cases into the hole
"+" in the positive direction.

PLANE OF OPERATION

The table below lists which axes are involved with the words X and Y to define the pocket dimensions in each of the three main planes.

	XY-PLANE	XZ-PLANE	YZ- PLANE
X-word parallel to	X-axis	X-axis	Z-axis
Y-word parallel to	Y-axis	Z-axis	Y-axis
Z-word (tool axis)	Z-axis	Y-axis	X-axis

The X and Y-word are programmed without sign.

FEEDRATE TO CUTTING DEPTH (F2=)

Feedrate to cutting depth (F2=) is the feed of the depth movement in the toolaxis for each cleaning pass.

If the Feed F2= is not defined in a cycle the Feed to the depth (K-word), is half the programmed feed (F).

CUTTING WIDTH IN % (I)

The value of the I-word states the percentage of the tool diameter to be used as cutting width for each cutting pass.

For example: I75 states that the cutting width is equal to 75% of the tool diameter.

If the I-word is not programmed, the value of a machine constant (MC720) is used.

CUTTING DEPTH (K)

If the pocket can not be cleaned out at depth in one pass, the K-word is used to program the depth for each cleaning pass.

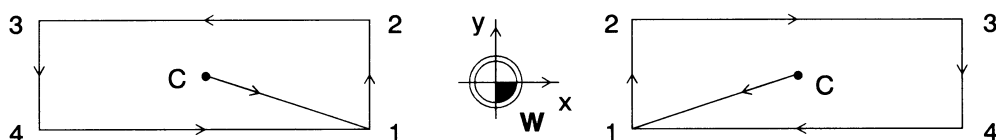
If the K-word is not programmed or has the same value as the Z-word, the groove is milled in one pass.

SPECIAL RETRACT DISTANCE (Y3=)

The special retract distance Y3= is added to the clearance distance (B-word) this gives the opportunity to avoid objects. The special retract distance can have a sign, which will normally be the positive one.

If the special retract distance (Y3=) is not defined a retract movement is executed to the clearance distance.

DIRECTION OF MILLING (J)



NB6075

J+1 Climb milling

J-1 Conventional milling

If no direction of milling is programmed, climb milling is assumed (J1 = default direction).

EXECUTION

A defined fixed cycle is executed on the position(s) programmed in either the G77 or G79 block. In these blocks the coordinates of the entry point of the pocket are stated.

The pocket is milled in the active plane (G17, G18 or G19). The radius of the actual tool stored in the tool memory, is used for milling the pocket.

Cycles must be programmed in G40 mode.

ROTATED POCKET

If the pocket is to be milled making an angle with the main axes, the G77 and G79 block are extended with a special word (B1=) to indicate the angle of rotation. Refer to the functions G77 and G79 ROTATED POCKET OR GROOVE for additional information.

Note: In the cycle the pocket is defined as if it is parallel to the axes.

CANCELLATION

The cycle's values are cancelled when a new cycle is defined or by softkey CLEAR CONTROL, M30 or by CANCEL PROGRAM.

FINISHING THE POCKET

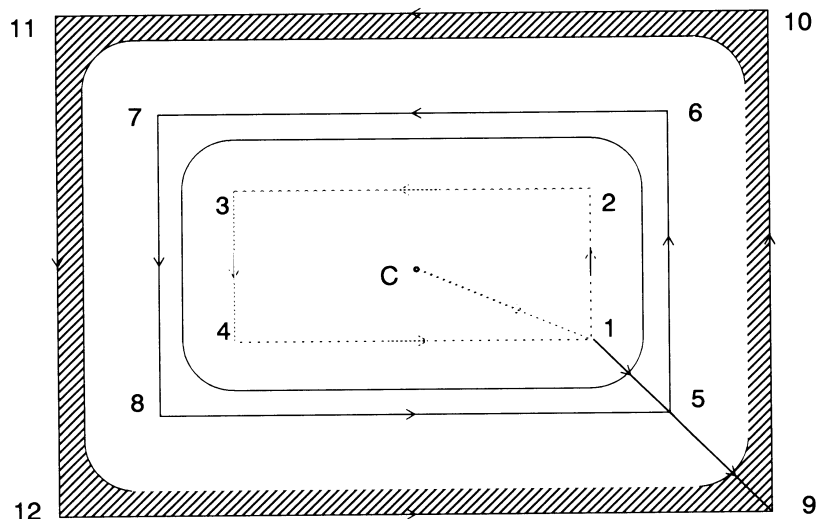
If a pocket must be milled and must have a finishing pass the following method may be used:

1. Add the allowance of the finishing cut to the tool radius and store the larger radius in the tool memory.
2. Execute the G87 milling cycle. A contour is milled which is smaller than the programmed contour due to the difference between the tool memory radius and the actual radius.
3. Program the contour of the pocket with the regular G1 and G2/G3 functions and execute the finishing pass with radius compensation and use the actual tool diameter.

THE TOOL SEQUENCE

The tool sequence for milling the pocket is:

- a. With rapid traverse to the centre (C) of the pocket and stay the clearance distance (B-word) above the workpiece.
- b. With half the programmed feed to the first depth (K-word).



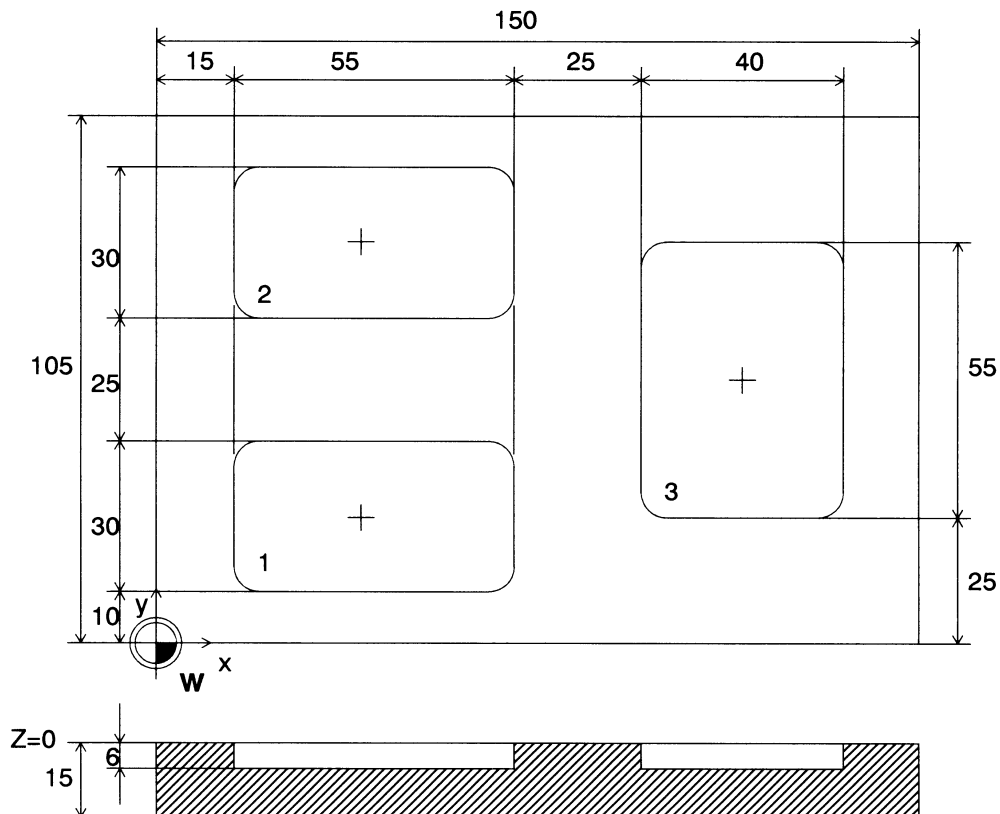
NB6076

Pocket milling sequence

- c. Move the tool from the centre to point 1 and mill around from 1 to 2, 3, 4 and back to 1. Point 1 is calculated by the control and depends on the X-word, Y-word and the radius of the active tool.
- d. Move the tool to point 5. The point is calculated by the control. The distances parallel to the axes are: I-word x tooldiameter
- e. Move the tool around from 5 to 6, 7, 8 and back to 5.
- f. Repeat the steps d and e -if necessary- until the layer is cleaned out.
- g. Finally follow the programmed contour and stop in the centre of the corner.
- h. If the programmed depth is reached, retract the tool to the clearance.

- i. If the programmed depth is not reached, move, with three times the programmed feed, to the centre (C) of the pocket.
- j. Clean out another layer by repeating the steps b to i

After the cleaning out, a finishing for the sides of the pocket might be necessary. The best way to proceed is to store in the tool memory, for the actual tool, a radius being the stock removal greater than the actual radius of the tool. Once the cycle is totally executed this stock removal remains for finishing. Activating the tool radius compensation and using the regular G1 and G2/G3 blocks program the finishing of the pocket.



NB5810

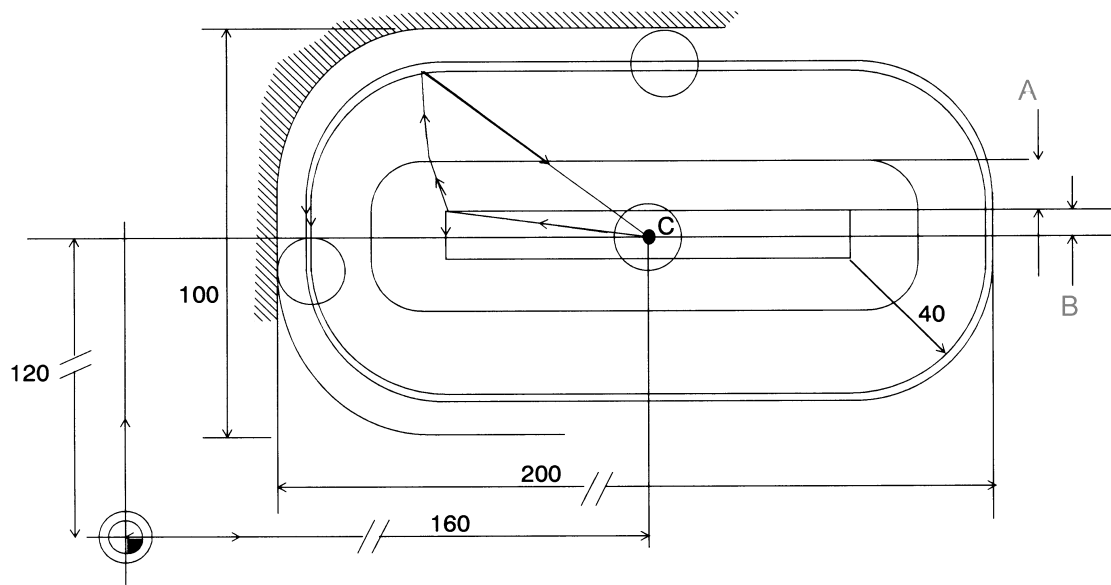
Milling three pockets.

Example with three pockets.

```

N10 T1 M6 (Fräser R5)
N20 G87 X55 Y30 Z-6 J+1 B1 I75 K1.5 F200 S500 M3
N30 G79 X42.5 Y25 Z0
N31 G79 X42.5 Y80 Z0
N40 G87 X40 Y55 Z-6 J+1 B1 I75 K1.5 F200 S500 M3
N50 G79 X115 Y52.5 Z0

```

Example

NB8639

A: 75% of the diameter
B: 75% of the radius

```

N10 T1 M6 (Mill R5)
N20 G87 X200 Y100 Z-6 J+1 B1 R40 I75 K1.5 F200 S500 M3
N30 G79 X160 Y120 Z0

```

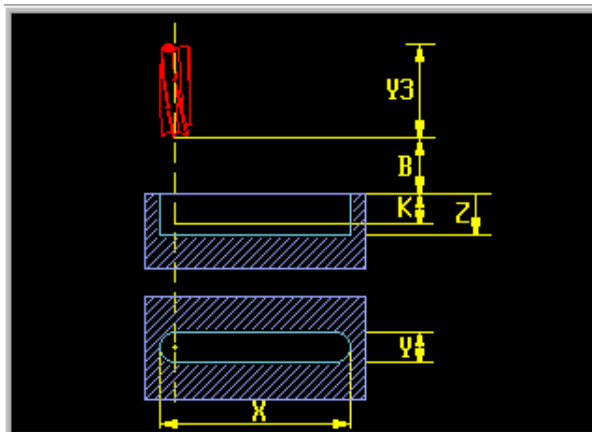
Explanation:

N10: Load tool 1.
N20: Define pocket milling cycle.
N30: Execute cycle at programmed position.

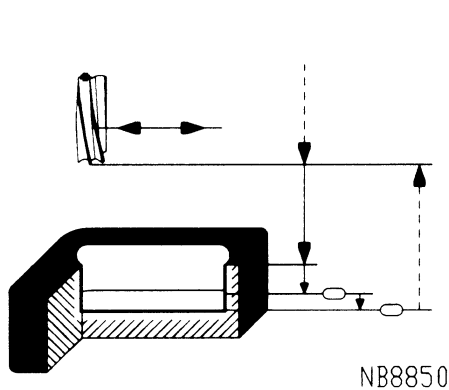
5.46 G88 Groove milling cycle

Purpose

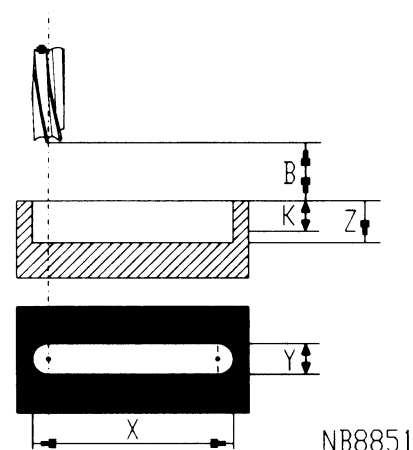
To define in one program block the geometry of a groove and some parameters for milling it.



Format



NB8850



NB8851

N... G88 X... Y... Z... {B...} {J...} {K...} {Y3=..} {F2=..}

Parameters

G	Groove milling cycle
X	Dimension parallel to X
Y	Dimension parallel to Y
Z	Total groove depth
B	Clearance
J	J1:climb / J-1:conventional
K	Cutting depth
Y3=	Special retract distance
F2=	In depth feed (only this block)

Groove geometry

X Dimension parallel to X
 Y Dimension parallel to Y
 Z Total groove depth
 F2= In depth feed (only this block)

Cutting parameters

B Clearance
 J J1: climbing / J-1: conventional
 K Cutting depth
 Y3= Special retract distance

Modal parameters

F, S, T, M, H, E...=
 T1=, T2=

Type of function

Modal

Associated functions

G77, G79, G81, G83-G87, G89

Notes and usage

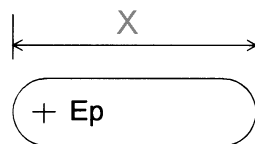
TOTAL DEPTH (Z)

Total depth of the groove measured from the surface.

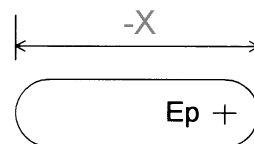
The sign of the Z-word indicates the direction of depth movement in the tool axis:

"_" in the negative direction, in most cases into the hole

"+" in the positive direction.



X... (G17), X... (G18), Y... (G19)



X-... (G17), X-... (G18), Y-... (G19)

(NB8683)

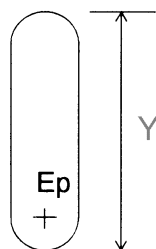
GROOVE PARALLEL TO X-AXIS

If the axis of the groove should be parallel to the X-axis, the programming is as follows:

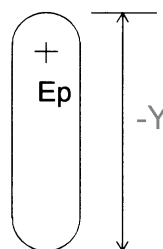
Absolute value of X-word > than value of Y-word

The sign of the X-word determines on which side of the entry point (Ep) the groove is milled.
 The Y-word is programmed without sign.

GROOVE PARALLEL TO Y-AXIS



Y... (G17), Y... (G18), X... (G19)



Y... (G17), Y... (G18), X... (G19)

(NB8683)

If the axis of the groove should be parallel to the Y-axis, the programming is as follows:
absolute value of Y-word > than value of X-word

The sign of the Y-word determines on which side of the entry point (Ep) the groove is milled.
The X-word is programmed without sign.

PLANE OF OPERATION

The table below lists which axes are involved with the words X and Y to define the length and width of a groove in the three main planes.

	XY-PLANE	XZ-PLANE	YZ-PLANE
X-word parallel to	X-axis	X-axis	Z-axis
Y-word parallel to	Y-axis	Z-axis	Y-axis
Z-word (tool axis)	Z-axis	Y-axis	X-axis

FEEDRATE TO CUTTING DEPTH (F2=)

Feedrate to cutting depth (F2=) is the feed of the depth movement in the Toolaxis for each cleaning pass.

If the Feed F2= is not defined in a cycle the Feed to the depth (K-word), is half the programmed feed (F).

CUTTING DEPTH (K)

If the groove cannot be cleaned out at depth in one pass, the K-word is used to program the depth for each cleaning pass.

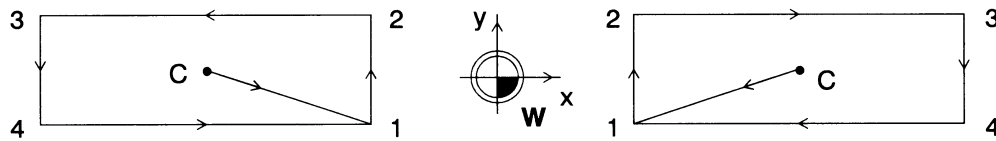
If the K-word is not programmed or has the same value as the Z-word, the groove is milled in one pass.

SPECIAL RETRACT DISTANCE (Y3=)

The special retract distance Y3= is added to the clearance distance (B-word) this gives the opportunity to avoid objects. The special retract distance can have a sign, which will normally be the positive one.

If the special retract distance (Y3=) is not defined a retract movement is executed to the clearance distance.

DIRECTION OF MILLING ON THE FINISHING PATH (J)



NB6075

J+1 Climb milling

J-1 Conventional milling

If no direction of milling is programmed, climb milling is assumed (J1 = default direction).

EXECUTION

A defined fixed cycle is executed on the position(s) programmed in either the G77 or G79 block. In these blocks the coordinates of the entry point of the groove are stated.

The groove is milled in the active plane (G17, G18 or G19). The radius of the actual tool stored in the tool memory, is used for milling the groove.

Cycles must be programmed in G40 mode.

ROTATED GROOVE

If the groove is to be milled making an angle with the main axes, the G77 and G79 block are extended with a special word (B1=) to indicate the angle of rotation. Refer to the functions G77 and G79 ROTATED POCKET OR GROOVE for additional information.

Note: In the cycle the groove is defined as if it is parallel to the axes.

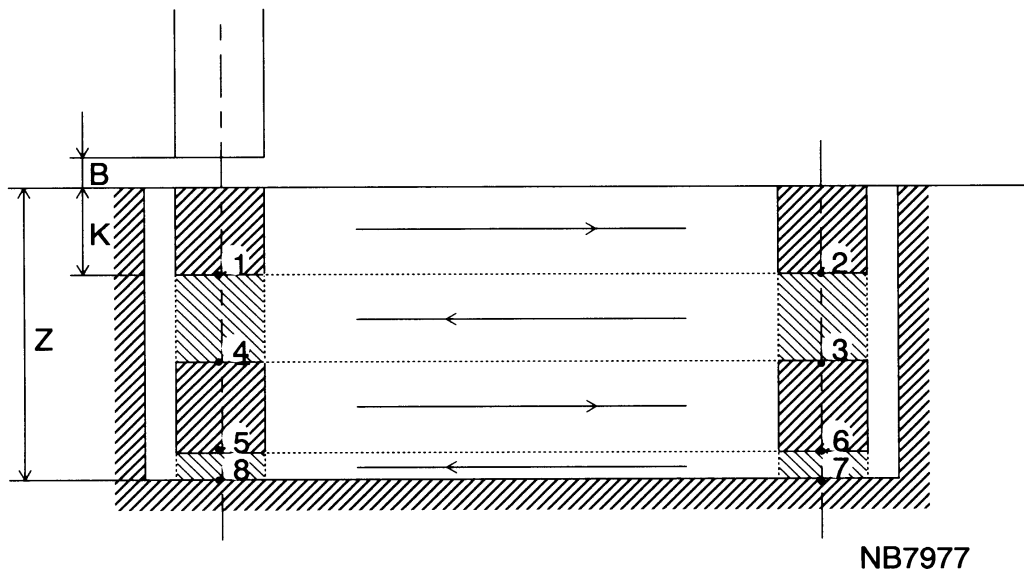
CANCELLATION

The cycle's values are cancelled when a new cycle is defined or by softkey CLEAR CONTROL, M30 or by CANCEL PROGRAM.

The tool sequence

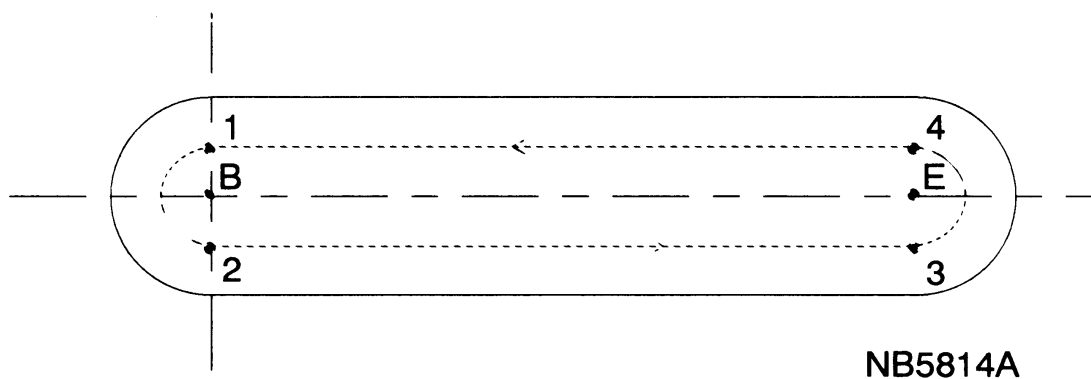
The tool sequence for milling the groove is:

- With rapid to point B where the tool enters the groove and stay the clearance distance (B-word) above the workpiece.
- With half the programmed feed to the first depth (1).
- With the programmed feed through the centre of the groove to point E (2).
- With half the programmed feed to the second depth (3).
- With the programmed feed through the centre of the groove back to point B (4).
- So the tool moves to and from, each time at another depth until the final depth is reached.



Depth movements in the groove.

- g. Once the final depth is reached, the sides of the groove are milled from B to 1, 2, 3, 4, 1, and back to B in a counter clockwise direction or a clockwise direction if J-1 is programmed. Here the tool radius compensation is automatically activated by the control and cancelled when the cycle is completed. The radius of the tool is used with the radius compensation.



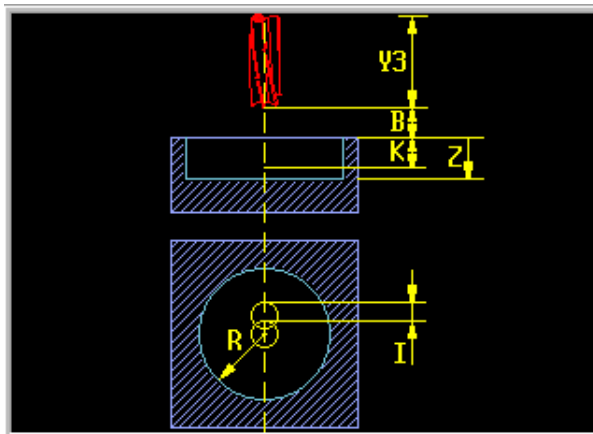
Cutter path for the sides.

- h. At the end of the cycle the tool is retracted out of the groove and stopped the clearance above the workpiece.

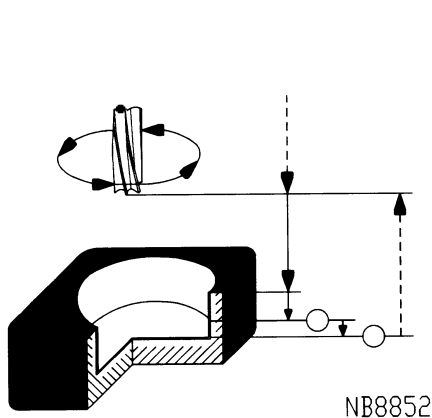
5.47 G89 Circular pocket milling cycle

Purpose

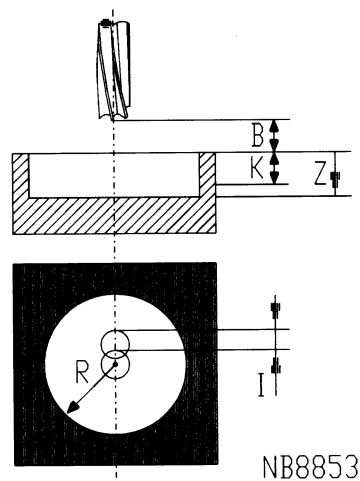
To define in one program block the geometry of a circular pocket and some parameters for cutting the pocket.



Format



NB8852



NB8853

N... G89 Z... R... {B...} {I...} {J...} {K...} {Y3=..} {F2=..}

Parameters

G	Circular pocket milling cycle
Z	Total pocket depth
B	Clearance
I	Cutting width mill in %
J	J1:climb / J-1:conventional
K	Cutting depth
R	Radius circular pocket
Y3=	Special retract distance
F2=	In depth feed (only this block)

Pocket geometry

Z Total pocket depth
 R Radius circular pocket
 F2= In depth feed (only this block)

Cutting parameters

B Clearance
 I Cutting width mill in %
 J J1: climbing / J-1: conventional
 K Cutting depth
 Y3= Special retract distance

Modal parameters

F, S, T, M, H, E...=
 T1=, T2=

Type of function

Modal

Associated functions

G77, G79, G81, G83-G88

Notes and usage**TOTAL POCKET DEPTH (Z)**

Total pocket depth measured from the surface. The sign of the Z-word indicates the direction of depth movement in the tool axis:

"-" in the negative direction, in most cases into the hole
 "+" in the positive direction.

PLANE OF OPERATION

The table below lists which axis in the active plane is used as the tool- AXIS.

	XY-PLANE	XZ-PLANE	YZ-PLANE
Z-word (tool axis)	Z-axis	Y-axis	X-axis

FEEDRATE TO CUTTING DEPTH (F2=)

Feedrate to cutting depth (F2=) is the feed of the depth movement in the toolaxis for each cleaning pass.

If the Feed F2= is not defined in a cycle the Feed to the depth (K-word), is half the programmed feed (F).

CUTTING WIDTH IN % (I)

The value of the I-word states the percentage of the tool diameter to be used as cutting width for each cutting pass.

For example: I75 states that the cutting width is equal to 75% of the tool diameter.

If the I-word is not programmed, the value of a machine constant is (MC720).

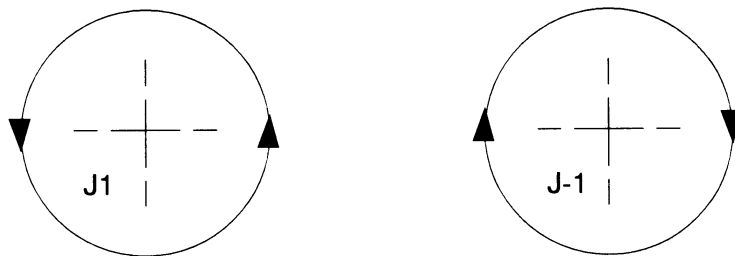
CUTTING DEPTH (K)

If the pocket cannot be cleaned out at depth in one pass, the K-word is used to program the depth for each cleaning pass.

If the K-word is not programmed or has the same value as the Z-word, the pocket is milled in one pass.

SPECIAL RETRACT DISTANCE (Y3=)

- The special retract distance Y3= is added to the clearance distance (B-word) this gives the opportunity to avoid objects.
- The special retract distance can have a sign, which will normally be the positive one. If the special retract distance (Y3=) is not defined a retract movement is executed to the clearance distance.

DIRECTION OF MILLING (J)

NB8554

J+1 Climb milling

J-1 Conventional milling

If no direction of milling is programmed, climb milling is assumed (J1= default direction).

EXECUTION

A defined fixed cycle is executed on the position(s) programmed in either the G77 or G79 block. In these blocks the coordinates of the centre of the pocket are stated.

The pocket is milled in the active plane (G17, G18 or G19). The radius of the actual tool stored in the tool memory, is used for milling the pocket.

Fixed cycles should always be programmed with the G40-mode.

CANCELLATION

The cycle's values are cancelled when a new cycle is defined or by softkey CLEAR CONTROL, M30 or by softkey CANCEL PROGRAM.

FINISHING THE POCKET

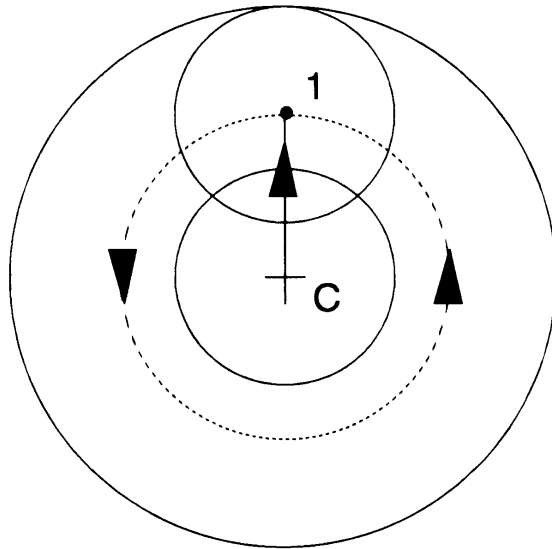
If a pocket must be milled and must have a finishing pass the following method may be used:

1. Add the allowance of the finishing cut to the tool radius and store the larger radius in the tool memory.
2. Execute the G89 milling cycle. A contour is milled which is smaller than the programmed contour due to the difference between the tool memory radius and the actual radius.
3. Program the contour of the pocket with the regular G1 and G2/G3 functions and execute the finishing pass with radius compensation and use the actual tool diameter.

TOOL SEQUENCE

The tool sequence for milling the circular pocket is:

- a. With rapid to the centre (C) of the pocket and stay the clearance distance (B-word) above the workpiece
- b. With half the programmed feed to the first depth (K-word)



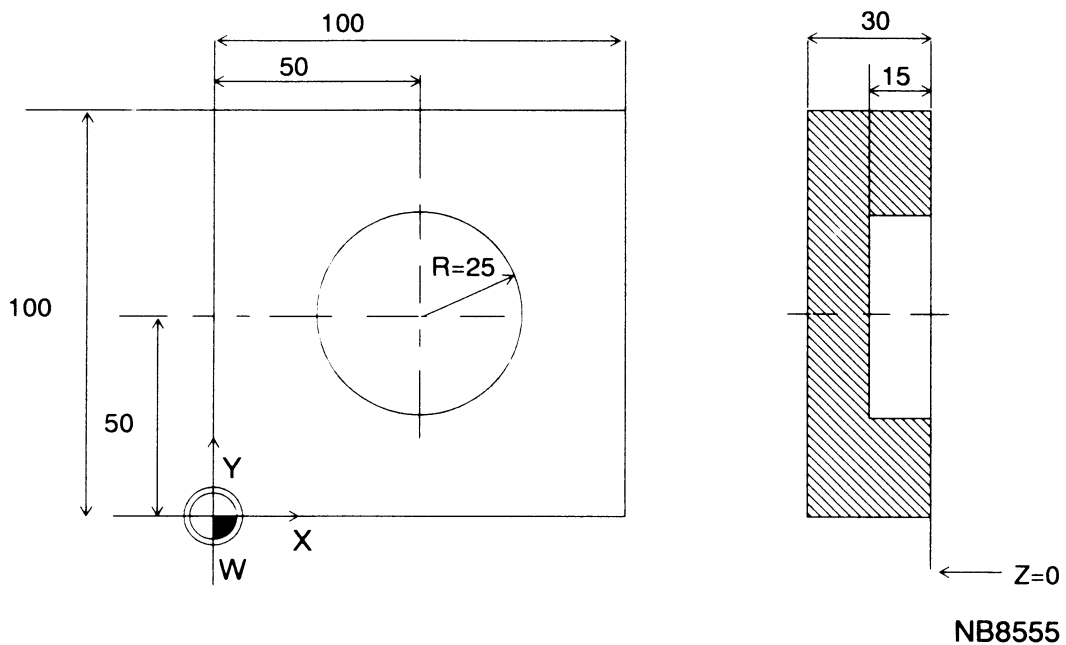
NB5817

Tool sequence for a circular pocket.

- c. With the programmed feed from C to 1. The distance to move is: tool diameter x I-word.
- d. Make with the programmed feed a full circle in clockwise direction (J-1) or counter clockwise direction (J+1) as seen from the tool.
- e. The steps c and d are repeated until all material is cleaned out from the first layer.
- f. Go with three times the programmed feed back to point C.
- g. If the programmed depth is not reached, another movement over the depth (K-word) takes place and then another layer is cleaned out.
- h. If the total depth is reached, retract the tool out of the pocket and stop the clearance above the workpiece.

After cleaning out, a finishing for the side of the pocket might be necessary. The best way to proceed is to store in the tool memory for the actual tool a radius, being the stock removal greater than the actual radius of the tool. Once the cycle is totally executed this stock removal remains for finishing. Activating the tool radius compensation and using the regular G1 and G2/G3-blocks program the finishing of the pocket.

Example



```

N10 T1 M6 (Mill R5)
N20 G89 Z-15 B1 R25 I75 K6 F200 S500 M3
N30 G79 X50 Y50 Z0
N40 G0 Z200

```

Explanation:

N10: Load tool 1.
 N20: Define the cycle for milling a circular pocket.
 N30: Execute cycle at programmed position.
 N40: Retract the tool

5.48 G90/G91 Absolute/incremental programming

Absolute and incremental programming can be done on two ways:

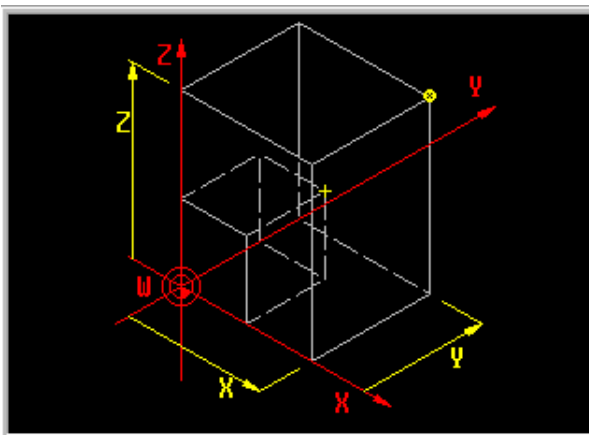
Absolute and incremental programming with G90 and G91.

Wordwise absolute and incremental programming with X90= and X91=.

5.48.1 G90/G91 Absolute/incremental programming

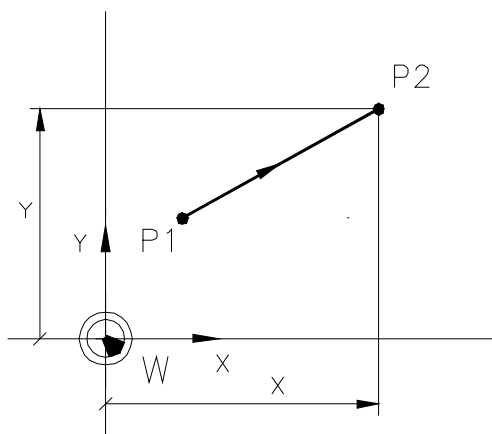
Purpose

To select one of two modes of coordinate programming.

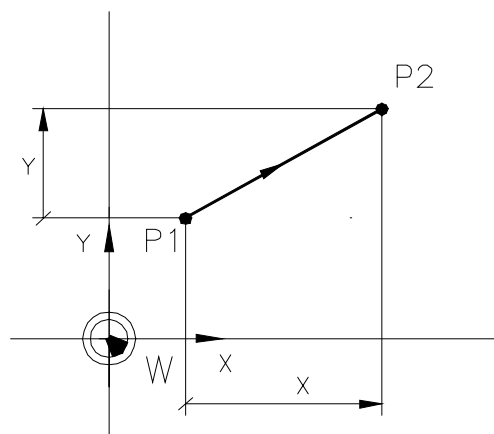


G90: Absolute coordinates measured from program zero point W.

G91: Incremental coordinates measured from last programmed tool position.



G90: Absolute coordinates



G91: Incremental coordinates

Format

N... G90/G91 [Axis coordinates]

Parameters

None

Possible axes coordinates by G90/G91 are:

X, Y, Z	Endpoint coordinates
A, B, C	End angels

Type of function

modal

Notes and usage

DEFAULT MODE

The G90 absolute coordinate mode automatically becomes active when the CNC system is switched on or at CLEAR CONTROL.

CANCELLATION

The G91 can be cancelled when G90 is defined or by softkey CLEAR CONTROL, M30 or by softkey CANCEL PROGRAM.

SWITCHING BETWEEN THE TWO MODES

The G90 default mode is cancelled by programming G91. The coordinates, which follow the switch, are interpreted by the CNC as incremental coordinates. To reactivate the absolute mode, G90 has to be programmed.

Internally the control operates with absolute coordinates. Therefore, within a particular program it is possible to change arbitrarily from absolute to incremental and vice versa.

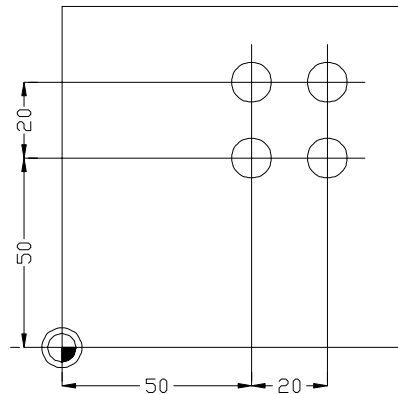
POLAR COORDINATES

The polar coordinates (B1=, L1=), (B2=, L2=), (B3=, L3=) are not influenced by the functions G90 and G91. They can be used arbitrarily in each coordinate mode.

POSITION DISPLAY

The axis positions on the display of the control are always absolute coordinates and related to the program zero point W.

Note: A part program should always contain an absolute position, before G91 is used. If a program starts immediately with G91, the actual tool position is used as the first absolute position of the program. Each time the program runs, this position should be the same, otherwise the program is executed each time at another place.

Example

```

N88550
N1 G17
N2 G54
N3 G195 X0 Y0 Z60 I100 J100 K-80
N4 S1300 T1 M6 (Drill R5)
N5 G0 X0 Y0 Z50
:
N8 G81 Y2 Z-10 F200 M3
N9 G79 X50 Y50 Z0
N10 G91
N11 G79 Y20
N12 G79 X20
N13 G79 Y-20
N14 G90
:
N17 G0 X0 Y0 Z50
N18 M30

```

Explanation:

N1 : Set the plane to be the XY-plane
 N2 : Set the zero point
 N3 : Graphic window definition
 N4 : activate Tool 1
 N5 : Move tool with rapid speed to start position
 N8 : Drilling cycle definition
 N9 : Drilling the first hole
 N10 : Switching to incremental mode
 N11 : A incremental movement to the second hole
 N12 : A incremental movement to the third hole
 N13 : A incremental movement to the third hole
 N14 : Switching to absolute mode
 N17 : Retract tool
 N18 : End of program

5.48.2 Wordwise absolute and incremental programming**Purpose**

Wordwise absolute and incremental programming, independently of G90/G91.

Format

Programming absolute:

N.. G.. [Axisname]90=...

Programming incremental:

N.. G.. [Axisname]91=...

Parameters

Axisname: X, Y, Z, U, V, W, I, J, K, A, B, C

X90=, Y90=, Z90=	Absolute endpoint
A90=, B90=, C90=	Absolute endpoint angle
X91=, Y91=, Z91=	Incremental endpoint
A91=, B91=, C91=	Incremental endpoint angle

Type of function

Non modal

Associated functions

G0, G1, G2, G3, G9, G45, G46, G61, G62, G77, G79, G145, G182

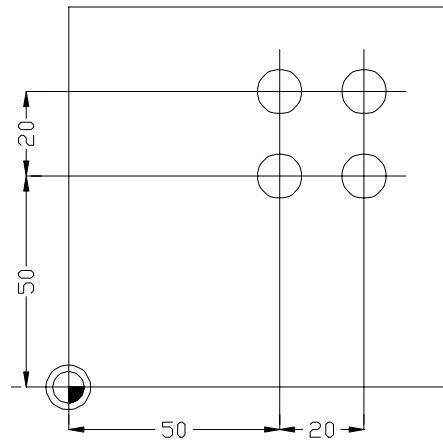
Notes and usage

Cartesian coordinates:

The wordwise absolute and incremental programming does not depend on the modally valid system of measurement 90/G91.

Polar coordinates:

Programming with polar coordinates will not be influence.

Example

```

N1 G17
N2 G54
N3 G195 X0 Y0 Z60 I100 J100 K-80
N4 S1300 T1 M6 (Drill R5)
N5 G0 X0 Y0 Z50

```

```

N8 G81 Y2 Z-10 F200 M3
N9 G79 X50 Y50 Z0
N11 G79 Y91=20
N12 G79 X91=20
N13 G79 Y91=-20
:
N17 G0 X0 Y0 Z50
N18 M30

```

Explanation:

```

N1 : Set the plane to be the XY-plane
N2 : Set the zero point
N3 : Graphic window definition
N4 : activate Tool 1
N5 : Move tool with rapid speed to start position
N8 : Drilling cycle definition
N9 : Drilling the first hole

N11 : Incremental Y movement to the second hole
N12 : Incremental X movement to the third hole
N13 : Incremental Y movement to the third hole

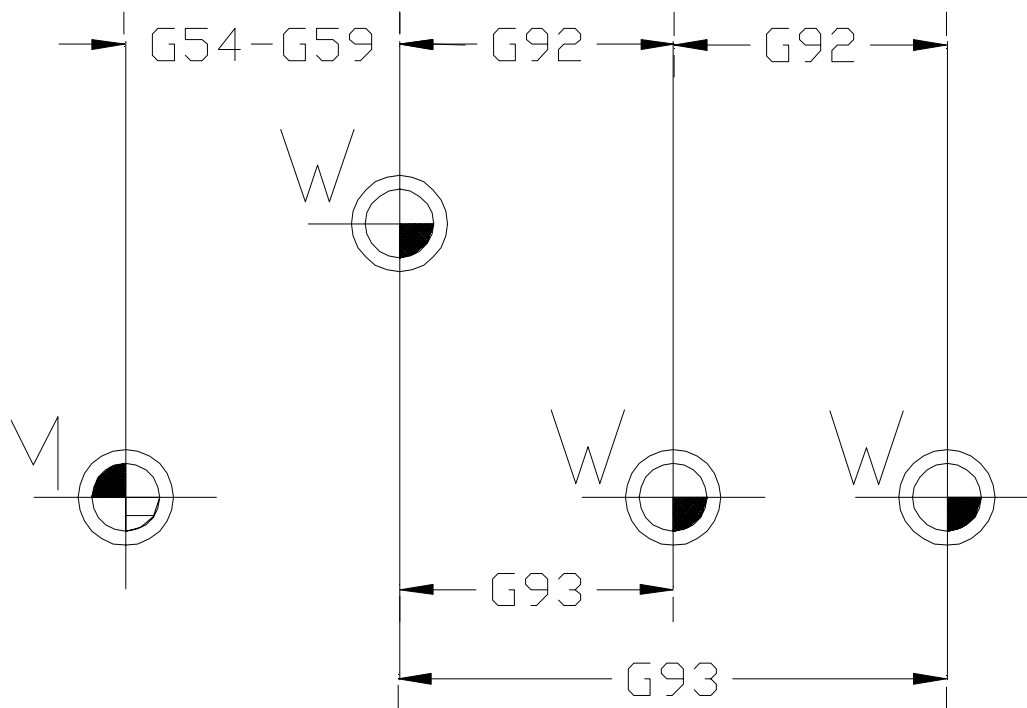
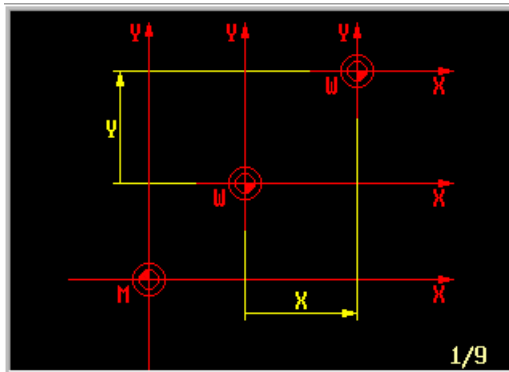
N17 : Retract tool
N18 : End of program

```

5.49 G92/G93 Incremental/Absolute zero point shift

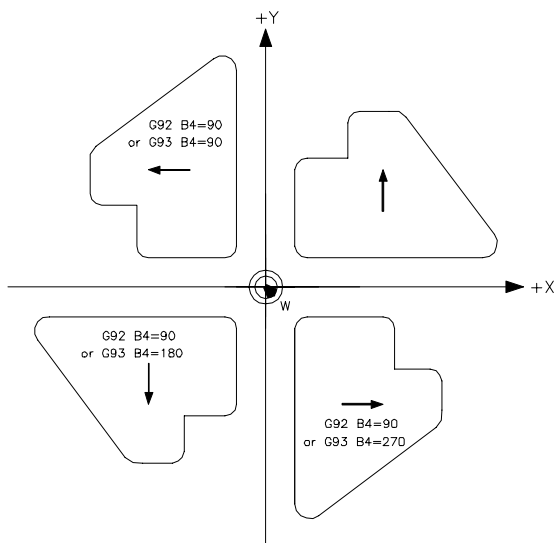
Purpose

1. To establish a program zero point (W) on the workpiece so that workpiece dimensions can be used directly for programming tool or workpiece movements. (G54-G59 or G54I[nr.])



- G92: Incremental zero point shift; shift values are related to last program zero point.
 G93: Absolute zero point shift; shift values are related to a fixed zero point

2. To produce a rotation of coordinate axes for a group of coordinates



G92: Rotation angle related to the last programmed main axis
 G93: Rotation angle related to a fixed machine tool.

Format

Zero point shift

N... G92 [Coordinate(s) related to last zero point]

N... G93 [Coordinate(s) related to a fixed zero point]

Axis rotation

N... G92/G93 B4=...

Parameters

G	Zero point shift abs./ rotation
X	Zero point coordinate
Y	Zero point coordinate
Z	Zero point coordinate
A	Zero point angle
B	Zero point angle
B2=	Polar angle
A3=	1: Reset position 0-360 degrees
B3=	1: Reset position 0-360 degrees
B4=	Angle of rotation absolute
L2=	Polar length
P1=	Point definition number

Words used with G92

1. Zero point shift

X, Y, Z	Zero point coordinate
A, B, C	Zero point angle
B1=	Angle
L1=	Path length
2. Axis rotation

B4=	Angle of rotation incremental
-----	-------------------------------

Words used with G93

1. Zero point shift

X, Y, Z	Zero point coordinate
A, B, C	Zero point angle
B2=	Polar angle
L2=	Polar length
P, P1=	Point definition number
2. Axis rotation

B4=	Angle of rotation incremental
-----	-------------------------------
3. Reset function

A3=, B3=, C3=	reset parameters
---------------	------------------

With G93 A3=1 the corresponding rotary axis position is reset to a value between 0 and 360 degrees. An A-axis with the position 370 degrees is changed after the programming of G94 A3=1 to 10 degrees.

Type of function

Non-modal.

Associated functions

With zero point shift:	G51/G52, G53-G59 or G54I
With axis rotation:	G72/G73

Notes and usage**ZERO POINT SHIFTS****INCREMENTAL ZERO POINT SHIFT (G92)**

The function G92 is used to shift the zero point from the actual program zero point (W) to a new program zero point (W).

APPLICATION OF G92

The function G92 is useful when programming identical tool movements, which are repeated at different locations on a workpiece.

ABSOLUTE ZERO POINT SHIFT (G93)

The function G93 is used to establish the program zero point by shifting the zero point from the mounting zero point (C) to the required program zero point (W).

UNALTERED AXIS

When a zero point shift is made and an axis is not involved, that axis does not need to be included in the block where the shift is programmed.

PROGRAMMED DIMENSIONS

All programmed dimensions which follow a zero point shift are measured from the new zero point (0,0).

DISPLAYED COORDINATES

Displayed axis coordinates are always related to the active program zero point W.

CANCELLATION OF G92 AND G93

With the G92 zero point shift is only the programmed addresses added to G93. With the G93 zero point shift is only the programmed addresses overwritten to a G92.

G93 X....

G92 Y.... Y is added to the G93 zero point shift.

A programmed zero point shift (G92 or G93) is cancelled if another zero point shift function (G51/G52, G53-G59 or G54I[nr.]) is programmed.

A programmed zero point shift (G92 or G93) is cancelled at end of program or by softkey CLEAR CONTROL, M30 or by softkey CANCEL PROGRAM.

G92/G93 IN TEACH IN MODE

A zero point shift, which is input in the TEACH-IN mode, is cancelled by CLEAR CONTROL or by the G51 to G59 functions. The G92/G93 function must therefore be included in the part program after the functions G51/G52 or G53-G59.

ROTATION OF AXES

The main plane axes can be rotated around the program zero point W. In this way a part program or a section of a part program can be rotated.

The programmed Coordinates refer to the rotated axes.

PLANE SELECTION

Axis rotation is performed in the active main plane, thus:

G17: X- and Y-axis are rotated

G18: X- and Z-axis are rotated

G19: Y- and Z-axis are rotated

ANGLE OF ROTATION

The angle of rotation is programmed with the word B4=.. The angle ranges from -360° to 360° and is measured as with polar coordinates.

INCREMENTAL ANGLE OF ROTATION (G92)

With G92 the angle is measured with the last active coordinate axis:

- with G17 or G18: the (rotated) X-axis

- with G19: the (rotated) Z-axis

ABSOLUTE ANGLE OF ROTATION (G93)

With G93 the angle is measured with the machine tool axis:

- with G17 or G18: the fixed X-axis

- with G19: the fixed Z-axis

ZERO POINT SHIFT AND AXIS ROTATION

In a G92 or G93 block a zero point shift and axis rotation is allowed. The order of execution is:

- first the zero point shift,
- then axis rotation.

The new zero point is the centre of rotation.

G51-G59 ZERO POINT SHIFTS

If one of the G-functions G51 to G59 is programmed after axis rotation, the function is executed in the non rotated axes.

MIRROR IMAGE AND SCALING

A combination of mirror image and/or scaling and axis rotation is allowed. The order of execution is:

- first scaling and mirror image,
- then axis rotation.

DISPLAYED COORDINATES

After axis rotation the displayed axis coordinates are related to the non rotated axes of the main plane.

CANCELLATION

The G92 axis rotation is cancelled, if a G93 axis rotation is programmed.

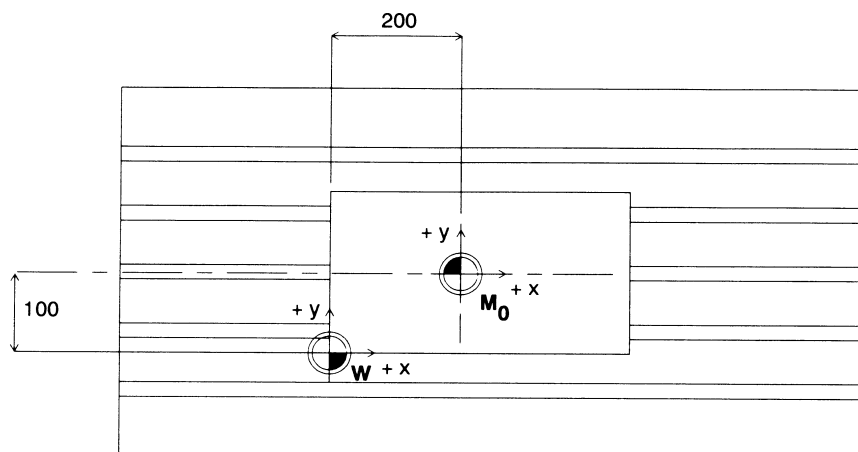
The G92 axis rotation is added to a G93 axis rotation.

The G93 axis rotation is cancelled with G93 B4=0.

Both axis rotations (G92 and G93) are cancelled at end of program or by softkey CLEAR CONTROL, M30 or by CANCEL PROGRAM.

Examples

Example 1.

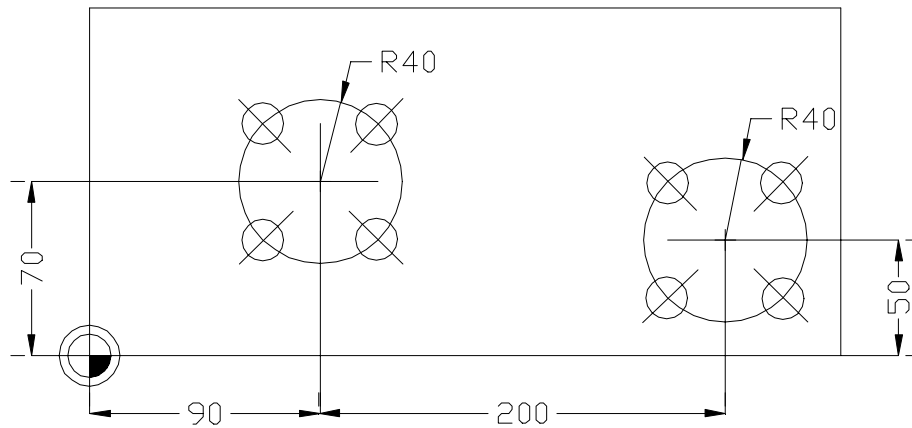


NB8556

In this example the centre of the workpiece coincides with the machine zero point (M₀), and the program zero point (W) is to be located in the left hand corner of the workpiece. A G93 can be used to set the program zero point:

N30 G93 X-200 Y-100

Example 2



The four holes around point A and the four holes around B are to be drilled. In the program the program zero point (W) is located at A or at B. In this way calculations during programming are reduced to a minimum.

Program with G92

```

N79560
N1 G17
N2 G54
N3 G195 X-10 Y-10 Z10 I420 J180 K-30
N4 G99 X0 Y0 Z0 I420 J160 K-10
N5 F200 S3000 T1 M6
N6 G92 X90 Y70
N7 G81 Y1 Z-12 M3
N8 G77 X0 Y0 Z0 I45 J4 R40
N9 G92 X200 Y-20
N10 G14 N1=8
N11 G92 X-290 Y-50
N12 G0 Z100 M30

```

Explanation:

N1 : Set the plane to be the XY-plane
 N2 : Set the zero point
 N3 : Graphic window definition
 N4 : Define the blank of the workpiece as a box
 N5 : activate Tool 1
 N6 : Incremental zero point shift
 N7 : Drilling cycle definition
 N8 : Drilling the four holes on the circle
 N9 : Incremental zero point shift
 N10 : Repeat programblock 8. Drilling the four holes on the circle
 N11 : Incremental zero point shift back to the first zero point shift
 N12 : End of program

Program with G93

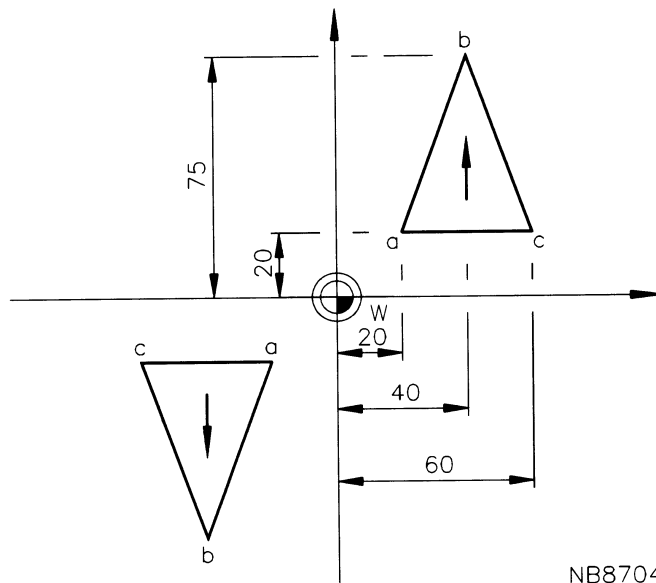
Related to the mounting point C, the program looks as follows:

```
N79561
N1 G17
N2 G54
N3 G195 X-10 Y-10 Z10 I420 J180 K-30
N4 G99 X0 Y0 Z0 I420 J160 K-10
N5 F200 S3000 T1 M6
N6 G93 X90 Y70
N7 G81 Y1 Z-12 M3
N8 G77 X0 Y0 Z0 I45 J4 R40
N9 G93 X290 Y50
N10 G14 N1=8
N11 G93 X0 Y0
N12 G0 Z100 M30
```

Explanation:

```
N1 : Set the plane to be the XY-plane
N2 : Set the zero point
N3 : Graphic window definition
N4 : Define the blank of the workpiece as a box
N5 : activate Tool 1
N6 : Absolute zero point shift
N7 : Drilling cycle definition
N8 : Drilling the four holes on the circle. The surface of the workpiece is define as Z=0.
N9 : Absolute zero point shift
N10 : Repeat programblock 8. Drilling the four holes on the circle
N11 : Absolute zero point shift back to the first zero point shift
N12 : End of program
```

Example 3. Rotation of axes



NB8704

N9300

```

N1 G17
N2 G54
N3 S400 T1 M6
N4 G0 X60 Y-10 Z1 M3
N5 G1 Z-16 F1000
N6 G43 Y20
N7 G41
N8 G1 X20
N9 X40 Y75
N10 X60 Y20
N11 Y-10
N12 G40
N13 G0 Z10
N14 G93 B4=180
N15 G14 N1=4 N2=12
N16 G0 Z100
N17 M30

```

Explanation:

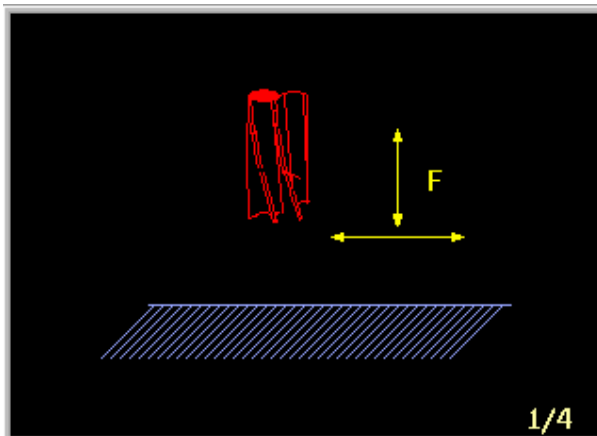
N1:	Set the XY-plane
N2:	Set the stored zero offset
N3:	Load tool 1 and set the spindle speed
N4:	Move tool to start position. Make spindle rotate clockwise at 400 rev/min
N5:	Move tool to depth at set feedrate
N6:	Move tool to point C
N7-N11:	Set radius compensation LEFT and cut workpiece
N12:	Cancel radius compensation
N13:	Move tool rapidly out of workpiece
N14:	Rotate axes through 180 degrees
N15:	Repeat instructions given in block N4 to N12
N16:	Retract the tool from the part
N17:	Cancel rotation of axes and end of program

5.50 G94/G95 Select feedrate unit

Purpose

To control how the CNC interprets programmed (F-word) feedrate values.

By turning mode refer to paragraph 12.5 "Extended select feedrate unit" on page 557.



G94	:	Feed rate in mm/min or inches/min.
G95	:	Feed rate in mm/rev or inches/rev.
G94 F5=	:	feed of rotary axes
		F5=0 degrees/min (default)
		F5=1 mm/min or inches/min

Format

N... G94/G95 F...

N... G94 F5=.

Parameters

```
G  Feed in mm/min (inch/min)
F  Feed
F1= F-adaptation:1=red.,2=r/h,3=high
F3= In depth feed
F4= In plane feed
F5= Feed rotary axes
```

G94

```
G  Feed in mm/rev (inch/rev)
F  Feed
F1= F-adaptation:1=red.,2=r/h,3=high
F3= In depth feed
F4= In plane feed
```

G95

Modal parameter

F, S, T, M

F1= F-adaption: 1=reduced, 2=reduced or high, 3=high.

F3= In depth feed

F4= In plane feed

Type of function

Modal

Notes and usage**DEFAULT MODE**

G94 is automatically made active when the CNC is switched on or by softkey CLEAR CONTROL, M30 or by CANCEL PROGRAM.

DIMENSIONAL UNIT

The dimensional unit for both functions is determined by the functions G71 (metric) or G70 (inches).

CONVERSION TO A FEEDRATE IN UNITS/MIN

When G95 is active the CNC automatically converts the F-value to a feed rate in mm/min (inches/min). If a spindle transducer is fitted to the machine, the measured spindle speed is used for this calculation.

MACHINES WITH KINEMATIC MODEL

The function G94 F5= is only possible if a kinematical model is defined for the machine. (MC312 must be active).

ROTARY AXIS RADIUS CALCULATION G94 F5=1

In machines with the kinematical model the rotary axis radius between the centre point of the rotary axis and of the workpiece can be calculated. This means that A40=, B40= and C40= no longer need to be programmed.

TURN OFF G94 F5=1

G94 F5=1 is cancelled by G94 F5=0, G95, the programming with A40=, B40= or C40= in G0 or G1, M30, <program abort> or <reset CNC>.

Example

N.. G94

N.. G1 X.. Y.. F200

The tool is moved to a point defined by the coordinates X.. and Y.. at a feed rate of 200 mm/min

N.. G95

N.. G1 X.. Y.. F.5

The tool is moved to a point defined by the coordinates X.. and Y.. at a feed rate of .5 mm/rev.

5.51 G96/G97 Constant cutting speed

Purpose

G96 Programming constant cutting speed.

G97 switches off constant cutting speed.

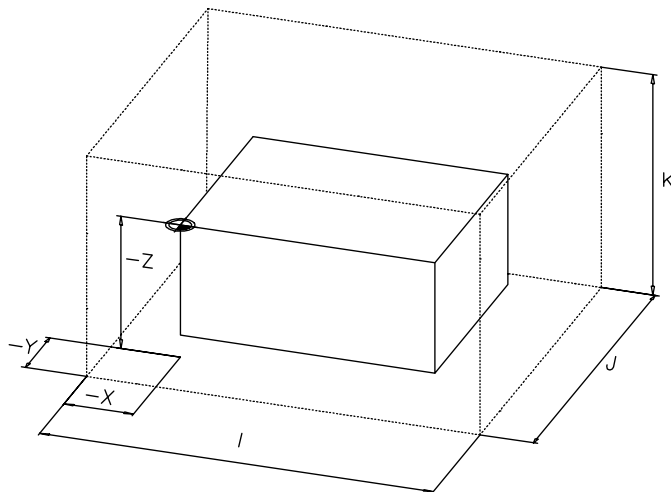
For explanation refer to paragraph 12.6 in chapter "Turning mode" on page 558

5.52 G98 Graphic window definition

It is recommended to start a new program always with G195.

Purpose

To define the position relative to the zero point W and the dimensions of a 3D graphics window in which the machining of a workpiece is to be represented with the graphical simulation of a part program run on the display of the control.



3D graphic window definition

Format

N... G98 X... Y... Z... I... J... K... {B...} {B1=...} {B2=...}

Parameters

G	Graphic window definition
X	Start point coordinate
Y	Start point coordinate
Z	Start point coordinate
B	Rotation around hor. axis (3D)
I	Dimension parallel to X
J	Dimension parallel to Y
K	Dimension parallel to Z
B1=	Rotation around vert. axis (3D)
B2=	Rotation around third axis (3D)

Type of function

Non-modal.

Associated functions

G99, G195 to G199

Notes and usage

GRAPHICAL SUPPORT

Refer to the appendix GRAPHICAL SUPPORT at the end of this manual for a short overview about the graphical support provided in the CNC PILOT control system and to the user manual for using the graphical support.

GRAPHIC WINDOW

The window, thus a bounded area on the display, is a rectangular 3D box which dimensions are defined by the G98-function. The dimensions of the workpiece are defined in a G99 block.

The window is used with the graphical simulation, but also with the synchron graphics with which the actual tool movements on the machine can simultaneously be seen on the display of the control.

DEFAULT WINDOW DIMENSIONS

If the 3D window dimensions are not defined the CNC uses the limit switches distances as default values.

TOOL IMAGE

A tool image can be assigned to the tool with the aid of the G-word in the tool memory. The required image can be selected from a set of available tool images and is used by the CNC system to accurately simulate the machining.
(For tool image refer to operation manual)

ANGLE OF VIEWING (B, B1=, B2=)

With the simulation graphics or the 3D-wire plot the workpiece can be seen rotated. The angles for viewing the rotated workpiece on the display are defined by the words B, B1= or B2=.

	B	B1=	B2=
	rotation about	rotation about	rotation about
XY-plane (G17)	X-axis	Y-axis	Z-axis
XZ-plane (G18)	Z-axis	X-axis	Y-axis
YZ-plane (G19)	Y-axis	Z-axis	X-axis

Other methods are available for selecting an angle of viewing and are described in the user manual.

DEFAULT SETTINGS FOR ANGLES OF VIEWING

If the angles of viewing are not programmed the following default settings are automatically used by the control: B60, B1=30, B2=0

Note: The function G98 must be programmed before G99.
It is recommended to start a new program always with G195.

Example

```
N9000
N1 G98 X-20 Y20 Z-25 I140 J-90 K85
N2 G99 X-15 Y15 Z-20 I130 J-80 K75
```

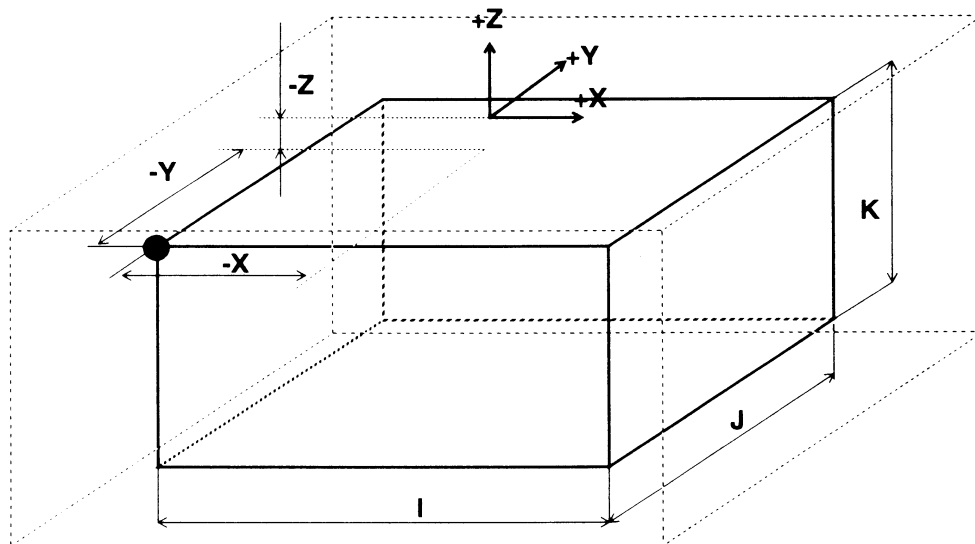
Explanation:

N1: Define the start point and dimensions of the 3D graphic window.
N2: Define the blank of the workpiece as a box.

5.53 G99 Definition of workpiece blank as a box

Purpose

To define the dimensions of a 3D box used as a blank (uncut) workpiece and the position of this 'blank' relative to the program zero point W. These dimensions are used in a graphical simulation of a part program run. (This function is used together with the G98-function).



NB8766

Format

N... G99 X... Y... Z... I... J... K...

Parameter

G	Graphic: material definition
X	Start point coordinate
Y	Start point coordinate
Z	Start point coordinate
I	Dimension parallel to X
J	Dimension parallel to Y
K	Dimension parallel to Z

Type of function

Non-modal

Associated functions

G98, G196 to G199

Notes and usage**IRREGULAR WORKPIECE SHAPE**

If the blank of the work piece cannot be defined with one box, the functions G196 to G199 must be used to define a workpiece, which has an irregular shape.

RESTRICTION

Only one G99 function is allowed in a part program.

Availability of the G99 function enables the part programs developed for the CNC 3000 to be used on the CNC Pilot as well. Unlike the CNC 3000, the program of the CNC Pilot can only contain a single G99 function. If a particular CNC 3000 program includes several G99 functions, the contour should be reprogrammed, using the functions G196 up to G199.

Note: The function G98 must be programmed before G99.

It is recommended to start a new program always with the functions G196 till G199.

Example

```
N9000
N1 G98 X-20 Y-20 Z20 I140 J90 K-85
N2 G99 X0 Y0 Z0 I100 J50 K-55
```

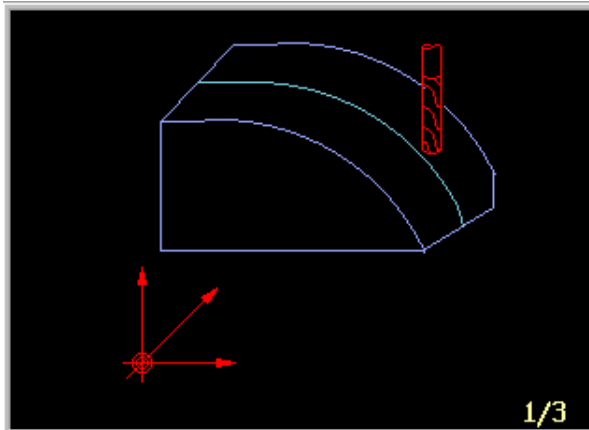
Explanation:

N1: Define the dimensions of 3D graphical window.
N2: Define the blank as a box

5.54 G141 3D-Tool correction with dynamic TCMP

Purpose

Permits the correction of tool dimensions for a 3D tool path that is programmed in these points by its end point co-ordinates and normalised vectors perpendicular to the surface.



Format

To activate 3D-tool correction

N.. G141 {R..} {R1 =..} {L2=}

To program straight-line movements

G141

N... G0/G1 [end point coordinates] [I.. J.. K..]

TCPM with active kinematical model

N... G0/G1 [end point coordinates] {I1=.. J1=.. K1=..} {A, B, C} {F..}

To delete 3D-tool correction

N.. G40

Parameter

```
G  3D tool correction
R  Nominal tool radius
L2= Rotary axes (0=shortest, 1=abs.)
R1= Nominal tool corner radius
```

With G141

R Nominal tool radius
 R1= Nominal tool corner radius
 L2= Circular axes (0=shortest, 1=absolute)

The R.. and R1=.. values should be the same as the nominal tool dimensions used by the programming system for calculating the toolpath. These values are set equal to zero, if not programmed.

With G0/G1

X, Y, Z	Linear end point coordinates
I, J, K	Axis components of surface normal vector.
I1=, J1=, K1= (TCPM)	Axis components of tool vector
A, B, C (TCPM)	Circular axis components of tool vector
F	Feed along the path

Modal words

F, S, some M-functions

Type of function

Modal

Associated functions

G40 and G412 to G44 for radius correction in a plane

For TCPM G8

General principles of G141

When milling a 3D surface, a given tool is moved along the surface in straight-line movements with a particular tolerance.

The calculation of the tool path on a 3D surface requires many calculations that are usually carried out by an NC programming system or a CAD system.

The calculated tool path depends on the shape of the tool, the dimensions of the tool and the tolerance to the surface.

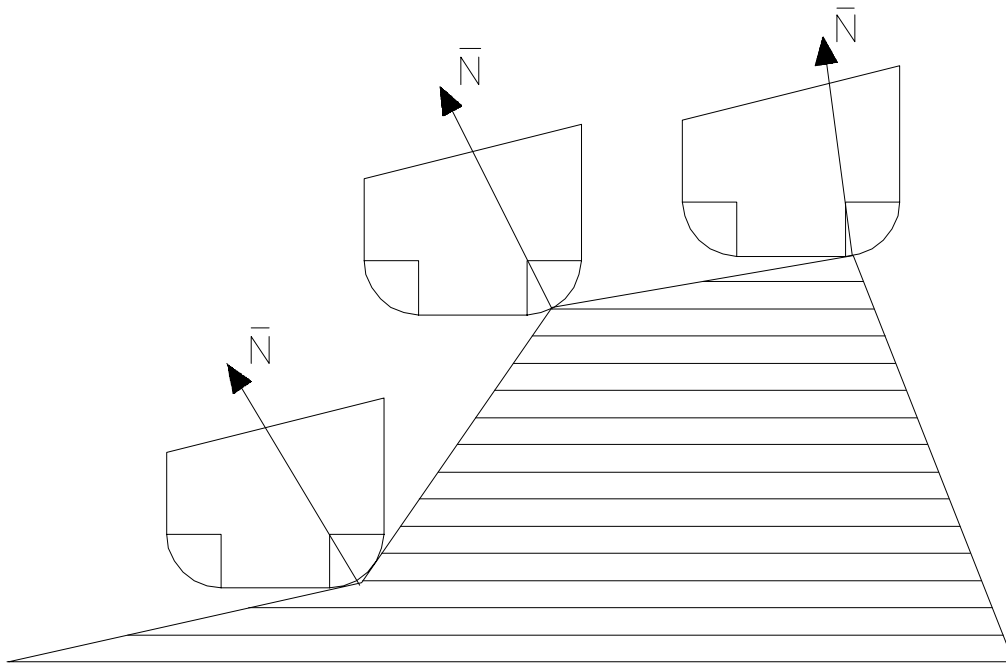
When executing the appropriate program **without G141**, the milling tool used must have the same dimensions as in the calculations, i.e. a standard milling tool must be used.

If a new tool is required while machining a 3D surface, this tool must also have the same dimensions as the **standard tool**.

If dimensional deviations are detected on the workpiece, a new calculation must be made using the programming system.

The 3D tool correction (**G141**) allows the use of tools whose dimensions differ from the dimensions of the standard milling tool. The corrections are carried out with the help of the direction vectors that are created by the programming system together with the end point co-ordinates.

In addition, the workpiece dimensions can be calculated by the programming system and the tool path by the CNC from the normalised vectors and the tool dimensions.



$\bar{N}$ = Surface normal vector (I, J, K)

Notes and application

The values of R.. and R1=.. should correspond to the nominal tool dimensions as used by the **programming system** for calculating the tool path. If these values are not programmed, they automatically become zero.

RADIUS (R, R1=)

R defines the tool radius with which the end points of the G0/G1 blocks are calculated in the CAD system.

R1= defines the tool corner radius with which the end points of the G0/G1 blocks are calculated in the CAD system.

General principles of TCPM

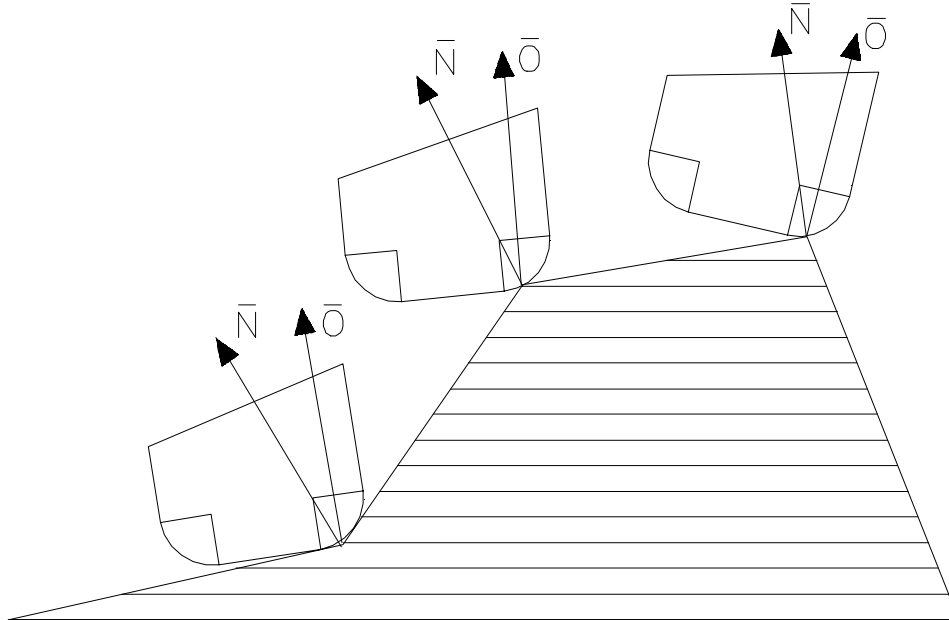
Maintaining position of tool tip when positioning swinging axes (TCPM)

(TCPM stands for "Tool Centre Point Management").

With **G141 '3D tool correction without TCPM'**, a curved (CAD) surface can be travelled taking the current tool dimensions into account. In this case, the path is described by end point co-ordinates and vectors perpendicular to the surface. The G141 function only guides the three linear axes but not the circular axes. In this way, the tool is always used in the same direction and is not guided over the workpiece surface at the optimum angle.

With **G8 'Tool orientation'** (static TCPM), the tool can be placed on the surface of the workpiece at an optimum angle. The G8 function is a feed movement and cannot be used continuously on a curved surface during a path movement.

In the case of **G141 with dynamic TCPM**, the tool is guided on a curved workpiece surface at an optimum angle. The current workpiece dimensions are taken into account. Dynamic TCPM is used for 5-axis milling. Dynamic TCPM also controls the circular axes. The tool is guided on the curved workpiece surface either vertically or at a programmed orientation.



$\bar{N}$ = Surface normal vector (I, J, K)

$\bar{O}$ = Tool vector (I1=, J1=, K1=) or Rotary axes coordinates of the tool vector (A, B, C)

The programming format of the linear blocks within G141 is expanded to include the option of programming a tool vector. Possible combinations are surface normal vectors and/or tool vectors. If only the tool vector is used, the tool correction must be calculated in the CAD system.

G7 may be active. In this case, the surface normal vectors and the tool vectors are defined in the G7 level.

Notes and application

ADDRESSES (R, R1=, L2=) (TCPM)

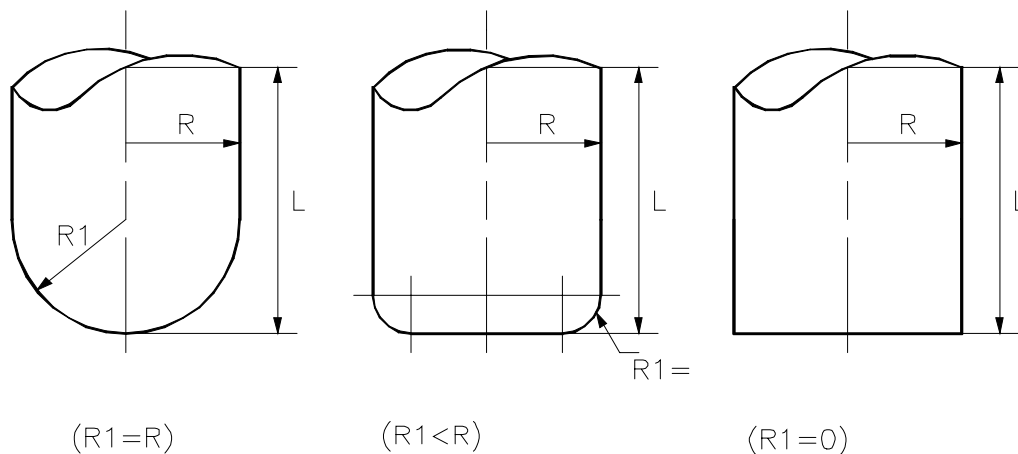
R defines the tool radius with which the end points of the G0/G1 blocks are calculated in the CAD system.

R1= defines the tool corner radius with which the end points of the G0/G1 blocks are calculated in the CAD system.

L2= 0 Circular axes travel the shortest distance (basic setting)

1 Circular axes travel to their absolute position (with circular axis programming).

POSSIBLE TOOLS



Tools used for the G141 function

TOOL MEMORY

The following dimensional details must be loaded into the tool memory to enable different types of tools to be used:

Radius milling tool	: R (tool radius), L (tool length), C (=tool radius)
Radius end milling tool	: R (tool radius), L (tool length), C (=rounding radius)
End milling tool	: R (tool radius), L (tool length), C0

If no value of C is entered, C automatically becomes 0.

The standard milling tool is thus an end milling tool.

Note: The rounding radius in the G141 block is programmed with the word R1=. The rounding radius is stored in the tool memory with the C word.

CREATED TOOL PATH

When the programming system creates the tool path (surface normal vector is programmed), the dimensions of the nominal tool (R.. and R1=) are programmed in the G141 block. The tool dimensions stored in the tool memory are used by the CNC to correct the tool path.

WORKPIECE DIMENSIONS

When the programming system creates the workpiece dimensions (surface normal vector and tool vector are programmed), the R.. and R1= words are not programmed in the G141 block. The tool dimensions stored in the tool memory are used by the CNC to calculate the tool path.

ACTIVATING G141

In the first block after G141, the milling tool travels from the current tool position to the corrected position in this block.

END POINT COORDINATES

Only absolute or incremental (X, X90, X91) Cartesian dimensional data can be used.

Up to V420, the co-ordinates in the first G141 block must be absolute and are measured from the programming zero point W.

G90/G91

The functions G90 and G91 are used for programming absolute (G90) or incremental (G91) dimensions. These functions must be alone in their own block.

MIRRORING

If the mirroring function (G73 and axis co-ordinates) is active before G141 is activated, the mirrored co-ordinates are used during the 3D tool correction.

Mirroring is possible as before once G141 is activated. Mirroring is cancelled by the G73 function.

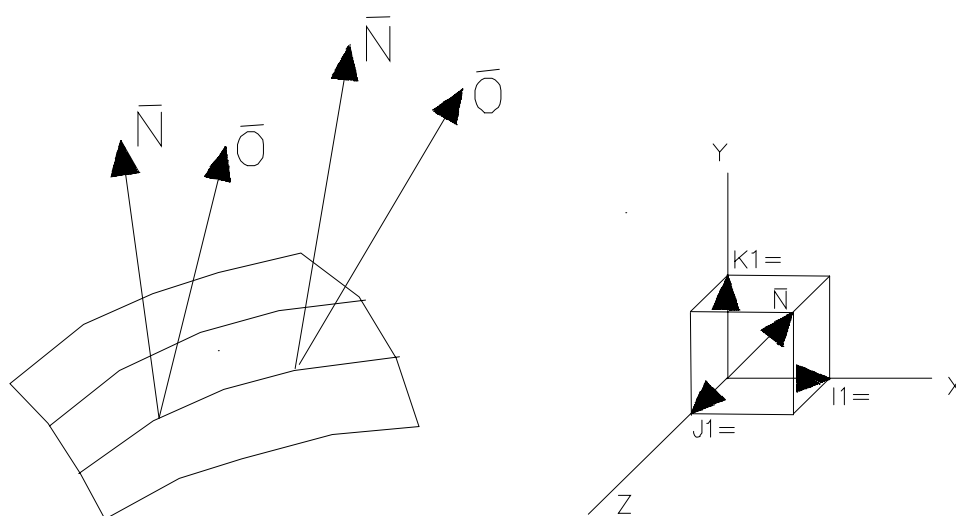
RADIUS CORRECTION G41...G44

After activating a G141 block, the effective radius correction programmed with G41...G44 is deleted.

SURFACE NORMAL VECTOR (I, J, K) (TCPM)

Defines the surface normal vector perpendicular to the surface.

The surface normal vector is perpendicular to the workpiece surface. The tool is positioned so that this vector always passes through the centre point of the tool corner rounding. This vector controls the positioning of the linear axes within G141.

**VECTOR COMPONENTS**

The vector components of the axes are independent of the level selected.

If no vector components are programmed in a block, the values last programmed are used.

If no components are programmed in the first block, the components not programmed are set at zero.

DIMENSION FACTOR

The input format of the vectors (I, J, K, I1=, J1=, K1= words) is limited to three decimal places. The surface normal and tool vectors do not, however, have to have the length 1. To increase the dimensional accuracy, the values in question can be multiplied by a dimension factor between 1 and 1000. With the factor 1000, for example, the input accuracy of the vector components is increased to six significant figures.

BACK CUTTING

Back cutting or collisions between tool and material at points not to be machined are not detected by the CNC.

KINEMATIC MODEL (TCPM)

The kinematic model is used for calculations within G141.

If no kinematic model is active (MC312 'Free machining level' = 0), G141 remains compatible with the G141 functions in older CNC versions.

TOOL VECTOR (TCPM)

I1=, J1=, K1= axis components of tool vector

or

A, B, C circular axis components of tool vector

The tool vector or the circular axis co-ordinates indicate the direction of the tool axis. The tool is turned so that it is parallel to this vector. This vector controls the positioning of the circular axes (and the associated compensation movement with linear axes) within G141.

DELETING

Function G141 is deleted by G40, M30, the program interrupt softkey or the CNC reset softkey. The milling tool stops at the last corrected position. The circular axes are not turned back automatically.

FUNCTIONS TO BE DELETED

When working with G141, functions G64, scale change (G73 A4=..), axis rotation (G92/G93 B4=..) and G182 must be deleted.

The following G functions are permitted if G141 **(TCPM)** is switched on:

Basic motions	0, 1, 7
Levels	17, 18
Program control	14, 22, 23, 29
Positioning feed	4, 25, 26, 27, 28, 94, 95, 96, 97
Radius correction	39, 40, 141
Zero points	51, 52, 53, 54, 92, 93
Geometry	72, 73
Co-ordinate measurement modes	70, 71, 90, 91
Graphics	195, 196, 197, 198, 199

If a G function that is not permissible is programmed, error message P77 'G function and Gxxx not permitted' is issued.

The following G functions are permitted if G141 **(TCPM)** is active:

Basic motions	0, 1
	Parameters of G0 and G1 are limited
	G0 without positioning logic
Program control	14, 22, 23, 29
Positioning feed	4, 25, 26, 27, 28, 94, 95, 96, 97
Radius correction	40, 141
	G40 switches G141 off
Zero points	51, 52, 53, 54, 92, 93
Geometry	72, 73
Co-ordinate measurement modes	90, 91

If a G function that is not permissible is programmed, error message P77 'G function and G141 not permitted' is issued.

PROGRAMMING LIMITATIONS

G functions that are not listed above may not be used.

Point definitions (P) and E parameters may not be used.

No tool change may be made after activating G141.

Notes and application for TCPM**RISK OF COLLISION**

When G141 is switched on, compensation movements similar to those in G8 may occur.

In the case of the switch-on movement, the tool tip must not be resting on the surface of the workpiece and should be programmed with a distance from the material at least equal to the tool diameter.

Remark: If G141 is switched off via G40, M30 or program cancel, there is no compensation movement and the circular axes remain in their last positions.

When approaching the contour, it may happen that the table with the workpiece is turned through 180 degrees to achieve the programmed tool direction. **ATTENTION! RISK OF COLLISION!**

UNDERCUTTING

If the tool direction changes within a G1 block, this tool direction change is carried out interpolating with the movement to the end point. In doing this, the path between the start and end points is corrected for undercutting.

Undercutting is not detected during block transitions. This undercutting should be corrected by inserting a block without an end point and with only one change of the tool vector by the CAD system. In this case, the tool turns about the tool contact point until the new tool direction is reached.

DISPLAY

When G141 is active, a yellow icon is displayed behind the tool number and the programmed G141 tool vectors (I1, J1, K1) can be seen in the machining status (on the G7/G8 positions).

Remark: If G7 and G141 are active at the same time, the G7 angle or vector can be seen.

A small 'p' at the bottom right, near the 'axis letters', shows whether the position of the tool contact point or the position is in machine co-ordinates. The display changes with the same softkey as with G7.

FEED RATE

The programmed feed rate applies to the contact point between the surface and the tool. The tool head may make other movements.

ERROR MESSAGES

P341 Tool vector incorrect

The tool vector (I1=, J1=, K1=) is incorrect. This error message is generated if all the components of the vector are zero.

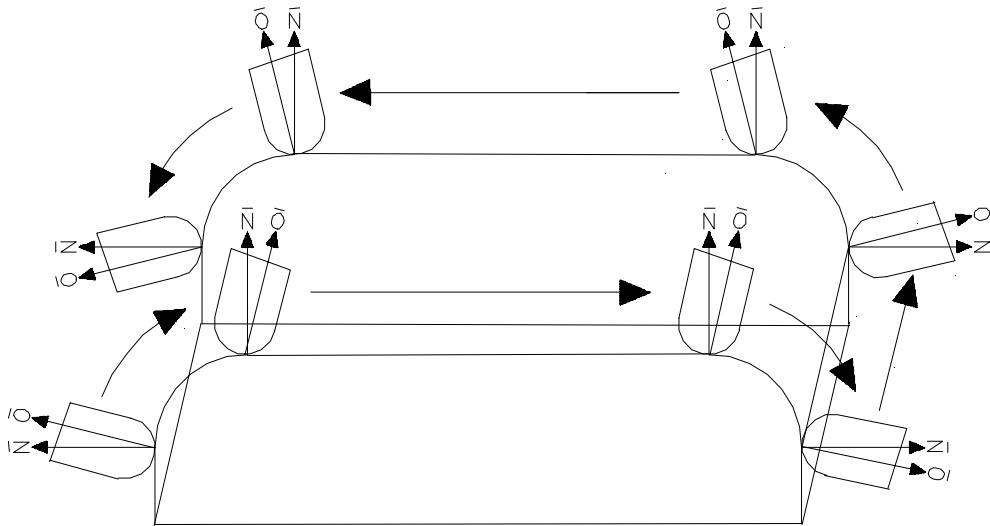
P342 Surface normal vector incorrect

The surface normal vector (I, J, K) is incorrect. This error message is generated if all the components of the vector are zero.

Example: G141 and TCPM

Tool vector with (I1=, J1=, K1=)

This program is independent of the machine.



N113 (square material with top rounding (R4) and swung tool (5 degrees))

N1 G17

N2 T6 M67 (10 round spherical milling tool: T6 R5 C5 in tool table)

N3 G54 I10

N4 G0 X0 Y0 Z0 B0 C0 S6000 M3

N5 F50 E1=0

N6 G141 R0 R1=0 L2=0 (all basic settings, do not need to be programmed)

N7 (R in CAD System is 0 mm)

N8 (R1 in CAD System is 0 mm)

N9 (L2=0 circular axes move shortest distance)

N10

N11 G0 X-1 Y=E1 Z0 I1=-1 K1=0

N12 (generated in CAD System)

N13 (front left arc)

N14 G1 X=0 Y=E1 Z=-4 I1=-0.996194698 K1=0.087155743

N15 G1 X=0.000609219 Z=-3.930190374 I1=-0.994521895 K1=0.104528463

N16 G1 X=0.002436692 Z=-3.860402013 I1=-0.992546152 K1=0.121869343

N17 G1 X=0.005481861 Z=-3.790656175 I1=-0.990268069 K1=0.139173101

N... (Each degree a point)

N100 G1 X=3.790656175 Z=-0.005481861 I1=0.034899497 K1=0.999390827

N101 G1 X=3.860402013 Z=-0.002436692 I1=0.052335956 K1=0.998629535

N102 G1 X=3.930190374 Z=-0.000609219 I1=0.069756474 K1=0.99756405

N103 G1 X=4 Z=0 I1=0.087155743 K1=0.996194698

N104 (front right arc)

N105 G1 X=36 Z=0 I1=0.087155743 K1=0.996194698

G-FUNCTIONS

N106 G1 X=36.06980963 Z=-0.000609219 I1=0.104528463 K1=0.994521895
N107 G1 X=36.13959799 Z=-0.002436692 I1=0.121869343 K1=0.992546152

N...

N194 G1 X=39.99756331 Z=-3.860402013 I1=0.998629535 K1=-0.052335956
N195 G1 X=39.99939078 Z=-3.930190374 I1=0.99756405 K1=-0.069756474
N196 G1 X=40 Z=-4 I1=0.996194698 K1=-0.087155743

N197 G40
N1971 (back right arc)
N1972 (move up to next cut)
N1973 G174 I100 (tool withdrawal)
N1974 G0 B0 C0 (rotate circular tables to original coordinates system)
N198 E1=E1+0.25
N1981 G1 Y=E1 (movement in normal X, Y, Z coordinates system)
N1982 G141

OR without deactivation G141

N197 (back right arc)
N198 E1=E1+0.25 (move up to next cut)

N199 G1 X=40 Y=E1 Z=-4 I1=0.996194698 K1=0.087155743
N200 G1 X=39.99939078 Z=-3.930190374 I1=0.994521895 K1=0.104528463
N201 G1 X=39.99756331 Z=-3.860402013 I1=0.992546152 K1=0.121869343

N...

N287 G1 X=36.13959799 Z=-0.002436692 I1=-0.052335956 K1=0.998629535
N288 G1 X=36.06980963 Z=-0.000609219 I1=-0.069756474 K1=0.99756405
N289 G1 X=36 Z=0 I1=-0.087155743 K1=0.996194698
N290 (back left arc)
N291 G1 X=4 Z=0 I1=-0.087155743 K1=0.996194698
N292 G1 X=3.930190374 Z=-0.000609219 I1=-0.104528463 K1=0.994521895
N293 G1 X=3.860402013 Z=-0.002436692 I1=-0.121869343 K1=0.992546152

N...

N379 G1 X=0.002436692 Z=-3.860402013 I1=-0.998629535 K1=-0.052335956
N380 G1 X=0.000609219 Z=-3.930190374 I1=-0.99756405 K1=-0.069756474
N381 G1 X=0 Z=-4 I1=-0.996194698 K1=-0.087155743
N382 E1=E1+0.25

N383 G14 N1=10 N2=389 J40

N384 G40
N385 G174 I100 (tool withdrawal movement)
N386 G0 B0 C0 (rotate circular tables to original coordinates system)
N387 M30

Example: G141 and TCPM

Identical workpiece
 Tool vector with (A, B, C)
 This program is machine dependent.

This program is for a machine with on the table a B-Axes under 45°, with upon a C-axes.

N114 (Rectangle block with rounding on top (R4) and tilting tool position (5 degrees))

N1 G17

N2 T6 M67 (Ball cutter round 10: In tool table T6 R5 C5)

N3 G54 I10

N4 G0 X0 Y0 Z0 B0 C0 S6000 M3

N5 F50 E1=0

N6 G141 R1=0 L2=0 (all default, so not necessary to program)

N7 (R in CAD System is 0 mm)

N8 (R1 in CAD System is 0 mm)

N9 (L2=0 Rotary axes moves shortest way)

N10

N11 G0 X-1 Y=E1 Z0 B180 C-90

N12 (generated in CAD System)

N13 (front arc left)

N14 G1 X=0 Y=E1 Z=-4 B145.658 C-113.605

N15 G1 X=0.000609219 Z=-3.930190374 B142.274 C-115.789

N16 G1 X=0.002436692 Z=-3.860402013 B139.136 C-117.782

N17 G1 X=0.005481861 Z=-3.790656175 B136.191 C-119.624

N... (Each degree a point)

N100 G1 X=3.790656175 Z=-0.005481861 B2.829 C1

N101 G1 X=3.860402013 Z=-0.002436692 B4.243 C1.501

N102 G1 X=3.930190374 Z=-0.000609219 B5.658 C2.001

N103 G1 X=4 Z=0 B7.073 C2.502

N104 (front arc right)

N105 G1 X=36 Z=0 B7.073 C2.502

N106 G1 X=36.06980963 Z=-0.000609219 B8.489 C3.004

N107 G1 X=36.13959799 Z=-0.002436692 B9.906 C3.507

N...

N194 G1 X=39.99756331 Z=-3.860402013 B206.449 C108.384

N195 G1 X=39.99939078 Z=-3.930190374 B210.629 C111.170

N196 G1 X=40 Z=-4 B214.342 C113.605

N197 (back arc right)

N198 E1=E1+0.25 (now translation)

N199 G1 X=40 Y=E1 Z=-4 B145.658 C66.395

N200 G1 X=39.99939078 Z=-3.930190374 B142.274 C64.211

N201 G1 X=39.99756331 Z=-3.860402013 B139.136 C62.218

N...

N287 G1 X=36.13959799 Z=-0.002436692 B4.243 C-178.499

N288 G1 X=36.06980963 Z=-0.000609219 B5.658 C-177.999

N289 G1 X=36 Z=0 B7.073 C-177.498

N290 (back arc left)

N291 G1 X=4 Z=0 B7.073 C-177.498

N292 G1 X=3.930190374 Z=-0.000609219 B8.489 C-176.996

N293 G1 X=3.860402013 Z=-0.002436692 B9.906 C-176.493

N...

N379 G1 X=0.002436692 Z=-3.860402013 B206.449 C-71.616

N380 G1 X=0.000609219 Z=-3.930190374 B210.629 C-68.830

N381 G1 X=0 Z=-4 B214.342 C-66.395

N382 E1=E1+0.25

N383 G14 N1=14 N2=382 J40

N384 G40

N385 G174 L100 (Retract tool)

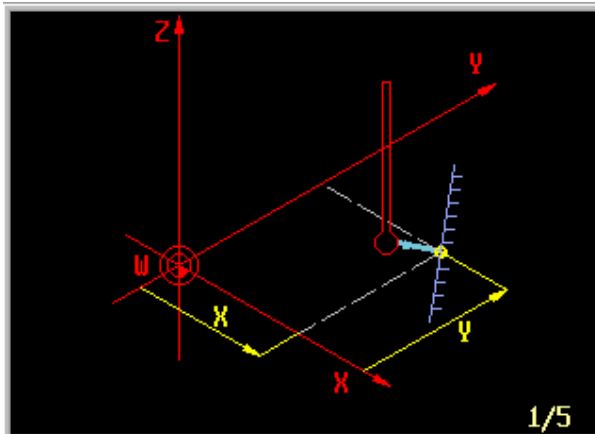
N386 G0 B0 C0 (turn rotary tables to original coordinates system)

N387 M30

5.55 G145 Linear measuring movement

Purpose

To execute a free programmable linear measuring movement to determine axis positions in measuring cycle macros.



Format

N... G145 [point to be measured] [(axis address) 7=...] {S7=...} E... {F2=...} {K...} {L...} {I3=...}
 N... G0 [axis coordinates]

Parameters

G Linear measuring movement
 X Endpoint coordinate
 Y Endpoint coordinate
 Z Endpoint coordinate
 A Endpoint angle
 B Endpoint angle
 K K0:tool correction on / K1:off
 L 0:measure by contact,1:by release
 E E-parameter for measure status
 B1= Angle
 B2= Polar angle
 X7= E-par. for measured value in X
 Y7= E-par. for measured value in Y
 Z7= E-par. for measured value in Z

A7= E-par. for measured value in A
 B7= E-par. for measured value in B
 ?90= Endpoint abs. (X,Y,Z...)
 ?91= Endpoint incr. (X,Y,Z...)
 I3= Status control (0=on, 1=off)
 L1= Path length
 L2= Polar length
 P1= Point definition number
 F2= Measure feed
 S7= E-par. for measured value in S

Measuring

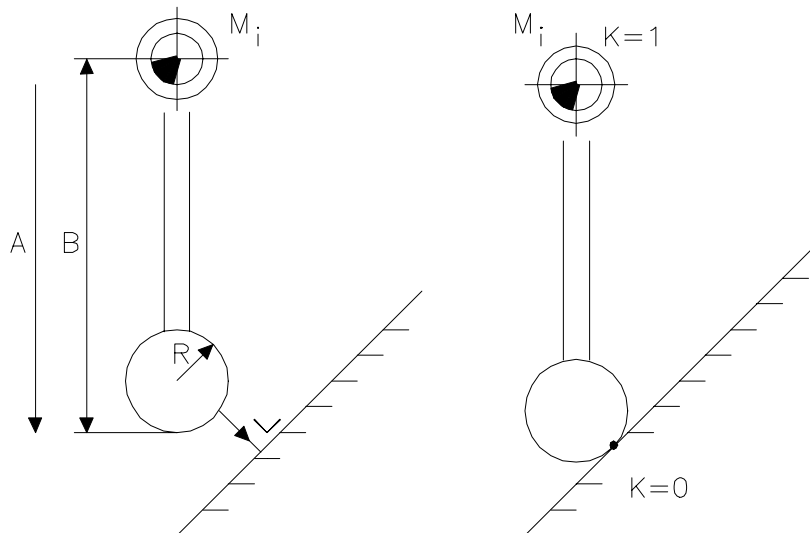
K	Tool correction	(0 = on	1 = off)
L	Measure	(0 = by contact	1 = by release)
E	E-parameter for measure status		
F2=	Measure feed		
I3=	Status control	(0 = on	1 = off)

Type of function

Non modal

Associated Functions

G148, G149, G150, G45, G46
M24, M26, M27, M28, M29

Notes and usage**COMPENSATING FOR PROBE DIMENSIONS (K)**

A: Tool axis
B: Tool length

K0: Tool correction on. Measured positions are corrected for the tool length and radius. Measured positions of rotary axes are not corrected for tool data.

K1: Tool correction off. Measured positions are not corrected. If K is not programmed, K0 is automatically activated.

The following assumptions are used when the measured positions are corrected for the probe dimensions.

- The probe is parallel to the tool axis.
- The probe is perfectly spherical.
- The probe movement is perpendicular to the surface being measured.

MEASURING CONDITIONS (L)

L0: Measure at contact.

L1: Measure at probe release from the measuring surface (see example).
If L is not programmed, L0 is automatically used.

STORAGE MEASURED VALUE [(Axis address)7=]

This word states the E-parameter number which is to contain the measured axis position; e.g. X7=2, states that X-axis measured value must be stored in parameter E2; X7=E1 (E1=5) means measured value is stored in E5.

For example: S7=... spindle angle will be store in E-parameter 5.

Note: Before S7=... must be programmed an oriented spindle stop (M19). Otherwise has the spindle angle no meaning.

MEASURING PROBES STATUS (E)

The measuring probe can be in one of three states after completion of a G145 block.

The assigned E-parameter can therefore have one of three states.

- E... = 0: the programmed position has been reached, but no measuring point has been found. The assigned E-parameters, which contain measuring values, remain unaltered.
- E... = 1: during the measuring movement a measuring point has been found. The measured position of the axes has been entered into the E-parameters.
- E... = 2: the G145 block was executed while Block Search, Testrun or the Demo mode was active.

STATUS CONTROL (I3=)

The status control of the measuring probe inside the G145 can be disconnecting for certain devices (laser). The laser has no signal. The default is zero.

MEASURING FEEDRATE (F2=)

If F2= is not programmed a default value stored in a machine constant (MC843) is used automatically.

Note: If a function for spindle direction (M03 or M04) is entered, this function will be suppressed and an error code output.

The function G145 is not permitted, when G182 is active.

BLOCK SEARCH

During BLOCK SEARCH the measuring movement is simulated. The E-parameters, in which the measured coordinates were to be loaded, remain unaltered. The signals from the measuring probe are ignored.

DEMO

During DEMO a movement is executed towards the programmed position. The programmed coordinates are loaded into the E-parameters. The signals from the measuring probe are ignored.

TESTRUN

During TESTRUN the measuring movement is executed with the test feed rate (a Machine Constant (MC741)) or simulated if TESTRUN is executed without movements. The programmed coordinates are loaded into the E-parameters. When the probe is triggered during a movement, the movement is aborted and a collision error generated.

GRAPHICS

In the GRAPHICS modes the measuring movements are simulated. The programmed coordinates are loaded into the E-parameters. The signals from the measuring probe are ignored.

Note: In all the mentioned operation modes, the E-parameter for the measuring probe status gets the value 2. By checking this parameter in the measuring macros it is possible to avoid using parameters, which do not contain measured values.

INTERVENTION

With INTERVENTION the G145-movement is treated as a G1-movement. The status of the probe should not be changed between the start point of the measuring movement and the point of interrupt. If the status was changed, an error message is displayed. An error message is also displayed, if the probe is triggered during the repositioning.

Examples**EXAMPLE 1:**

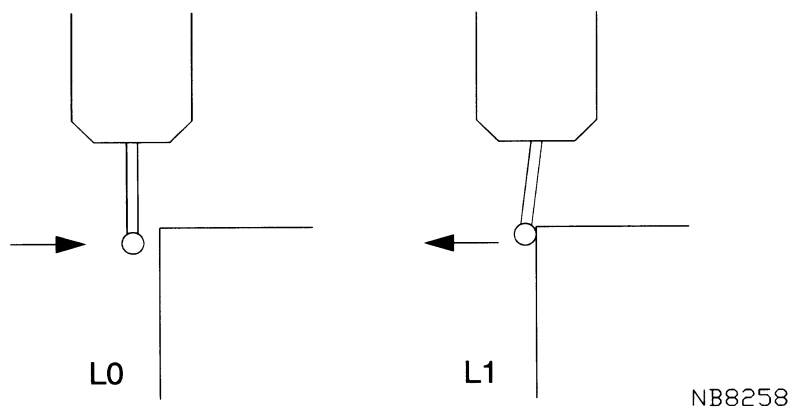
```

N90 G0 X25 Y18
N95 M27
N100 G145 X35 Y28 E12 F2=100 X7=25 Y7=26
N105 M28

```

Explanation:

N90: Start position linear measuring movement
 N95: Activate measure probe
 N100: Measure the point specified by X35, Y28 and store the result of the X-axis and Y-axis in E-parameter numbers 25 and 26 respectively. E-parameter number 12 stores the number indicating the probe status. F2=100 sets the measuring feedrate to 100 mm/ min.
 N105: Deactivate measure probe

Remark on programming with L1.

Two successive G145 blocks are to be programmed.

:

```

N21 G0 X35
N22 G28 I3=1
N23 G145 L0 X55 F2=2000
N24 G29 E0 E0=E7<1 N=26
N25 G145 L1 X45 X7=13 F2=500
N26

```

Explanations:

N21: Start position linear measuring movement
 N22: It is advisable to program the positioning function, in order that the switching point position is definitely reached.
 N23: Movement at high feed rate to the measuring surface. L0 must be programmed.
 N24: Check to see if the box is reached
 N25: The probe measures the programmed point when moving away from the surface. The measured X-coordinate is stored in parameter I3.

EXAMPLE 2: Measuring tool length with a measuring box

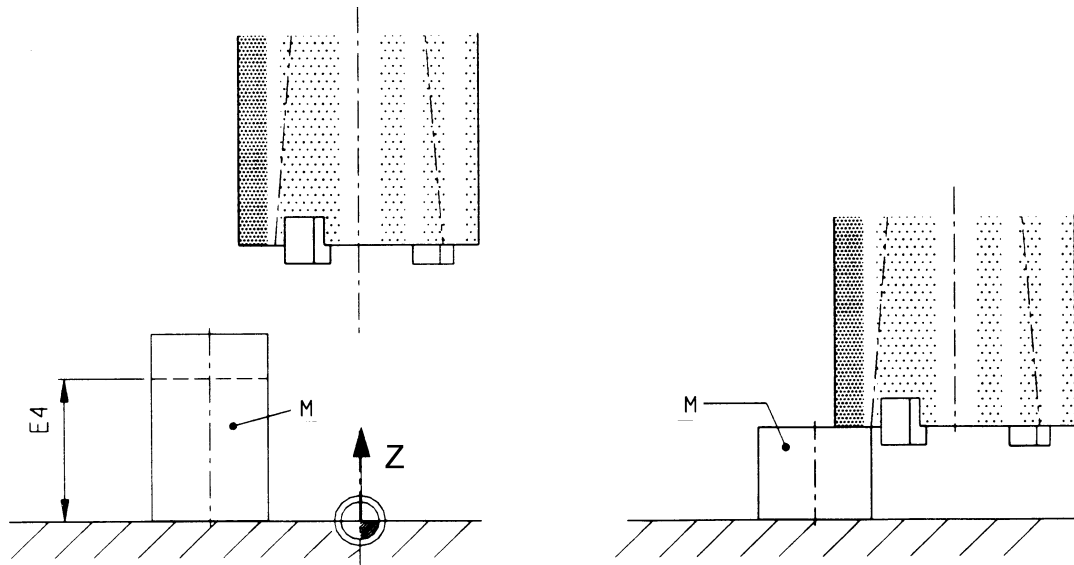
Two macros and a program are given for measuring the tool length with the aid of a measuring box.
 In the first macro (N14501) the trigger point of the box is determined.
 In the second macro (N14502) the actual length measurement is executed.
 In the program (N14503) parameters are set and both macros called.

Note

1. Refer to the machine tool builder's documentation to see if a measuring box can be used and which M-functions have to be used for switching on and off the measuring box in a particular installation.
2. The program and macros are given to show the possibilities of the measuring cycles and the E-parameters. Further update might be necessary to adapt the macros to the specific requirements of the user.

Used parameters

E0:	E-Parameter for jump function
E1:	X-coordinate of the measuring box
E2:	Y-coordinate of the spindle reference point
E3:	Z-coordinate of the measuring box
E4:	Y-coordinate of the trigger point of the box
E5	=0: Trigger point is not determined
	=1: Trigger point is already determined
E7	=0: No error found in macro N14501
	=1: An error found in macro N14501
E8:	Tool number or Tool identification number
E10:	Measured Y-coordinate

Macro for determination the trigger point of the measuring box.

M = Measuring box

E4 = Trigger point position

N14501 (Macro trigger position of the measuring box)

N1 T0 M6

N2 M24

N3 G0 X0 Y0 Z150

N4 G28 I3=1

N5 G145 Z20 L0 E7 F2=2000

N6 G29 E0=E7<1 E0 N=10

N7 G145 Z150 L1 E7 F2=200 Z7=4

N8 G0 Z150

N9 G29 E0 E0=1 N=12

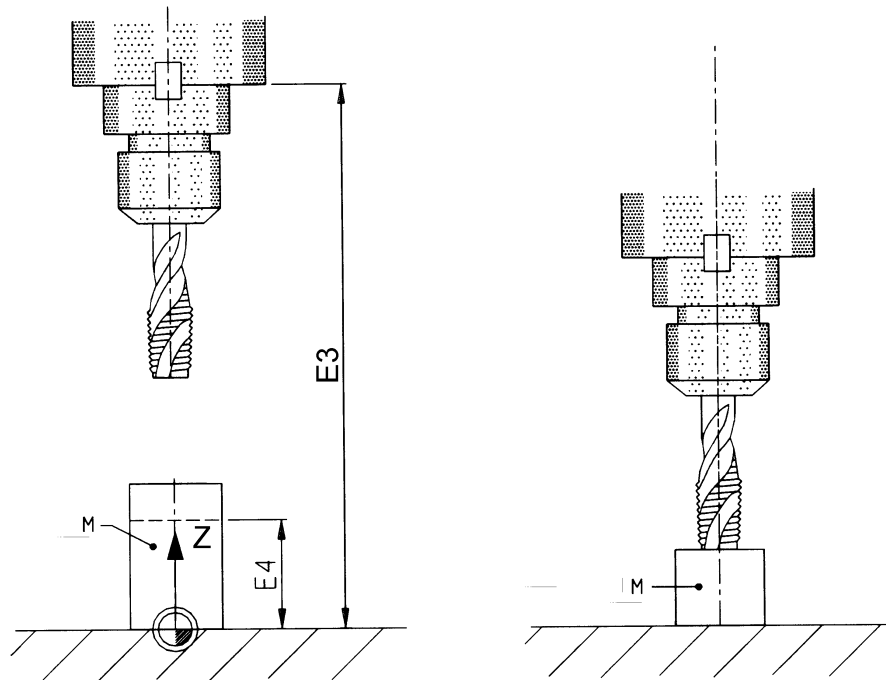
N10 M0

N11 (NO TRIGGER POINT FOUND)

N12

Explanation: of macro N14501

- N1: Unload the spindle
- N2: Activate the measuring box. M24 is an assumed function for activating the box.
- N3: Move the spindle nose above the measuring box
- N4: Feed movements should reach their position accurately.
- N5: Measuring movement in the Z-axis (=tool axis). A large Z-position is used so that it is sure that the box is reached. Measure at contact (L0).
- N6: Check to see if the box is reached. (E7=1!)
- N7: Measuring movement. Measure at release. At that moment the trigger point is found. Store measuring position in E4.
- N8: Retract the spindle
- N9: Jump to the end of the macro
- N10: Program stop
- N11: Message, that the trigger point is not found.

Macro for measuring the tool length.

M = Measuring box

E4 = Trigger point position

```

N14502 (Macro for measuring the tool length)
N1 T=E8 M6
N2 G0 X=E1 Y=E2 Z=E3
N3 G145 Z=(E4-20) L0 E7 F2=2000
N4 G29 E0=E7 <1 E0 N=10
N5 G145 Z=E3 L1 E7 Z7=10 F2=200
N6 G29 E0=E7 <1 E0 N=10
N7 G0 Z=E3
N8 G150 T=E8 L1=E10-E4
N9 G29 E0 E0=1 N=12
N10 M0
N11 (LENGTH MEASUREMENT UNSUCCESSFUL)
N12

```

Explanation: of macro N14502

- N1: Load the tool to be measured
- N2: Move the tool above the measuring box
- N3: Measuring movement in the Z-axis (=tool axis). A Z-position past the trigger point is used so that it is sure that the box is reached. Measure at contact (L0).
- N4: Check to see if the box is reached. (E7=1!)
- N5: Measuring movement. Measure at release. At that moment the length is found.
- N6: Check to see if a measurement took place (E7=1!)
- N7: Retract the tool
- N8: Update length of the actual tool in the tool memory
- N9: Jump to the end of the macro
- N10: Program stop
- N11: Message that tool length is not measured

Program for measuring tool length with a measuring box

The first time the program is used, the user has:

- to move the spindle nose above the measuring box
- to set the zero point with PRESET AXES
- to enter the parameters E5=0 and E8

If more tools have to be measured, parameter E5=1 has to be set and parameter E8 (the tool number) to be entered.

N14503 E8=.. (TOOL NUMBER)

N1 E5=0 (=0 DETERMINE TRIGGER POINT =1 NO TRIGGER POINT)

N2 G17

N3 G54

N4 G29 E5 K0 N=8

N5 G22 N=14501

N6 G29 E0 E0=E7<1 N=9

N7 E1=0 E2=0 E3=350

N8 G22 N=14502

N9 M28

N10 T0 M6

N11 G53

N12 M30

Explanation of the program

N1450

N1 Set the parameters E5 and E8

N2: Set the plane of operation to be the XY-plane

N3: Set the zero point found with PRESET AXES

N4: Check to see if the trigger point has to be determined (E5=0) or can be ignored.

N5: Call macro N=14501 to determine the trigger point of the measuring box

N6: Check to see if an error was found in macro N14501 (E7=1); in that case ignore the next macro call.

N7: Set the parameters containing the coordinates of the box

N8: Call macro N14502 to measure a tool length

N9: Switch off the measuring box. M28 is assumed to be used for that purpose.

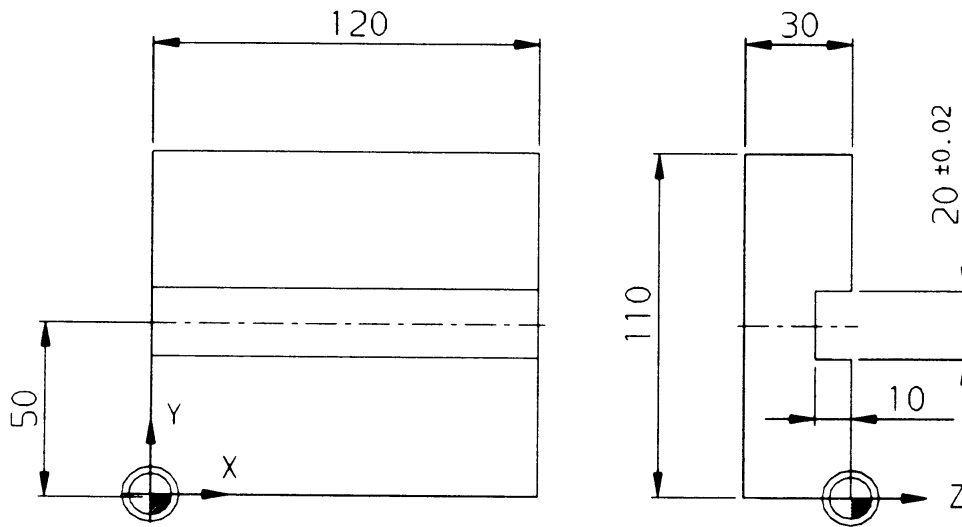
N10: Unload the spindle

N11: Restore the zero point

N12: End of the program

EXAMPLE 3: Milling and measuring a groove

Milling a groove followed by measuring the width of the groove. If the width is too small, the radius of the mill is changed and the groove is finished again.



```

N14504 (MILLING AND MEASURING A GROOVE)
N1 G17
N2 G54
N3 E15=20.02 (MAXIMUM WIDTH OF THE GROOVE)
N4 E16=19.98 (MINIMUM WIDTH)
N5 E3=(E15+E16):2
N6 T1 M6 (MILL Diameter 18 mm)
N7 G0 X-25 Y50 Z-10 F400 S1000 M3
N8 G1 X140
N9 G43
N10 G1 Y60
N11 G41
N12 X-25
N13 Y40
N14 X140
N15 G40
N16 Y50
N17 G0 Z50 M5
N18 G149 T0 E30
N19 T30 M6 (TOUCH TRIGGER PROBE)
N20 D207 M19
N21 M27
N22 X60 Y50 Z-8
N23 M29
N24 G145 Y65 E10 Y7=1 F2=500
N25 G0 Y50
N26 G29 E11=E10=0 E11 N=29
N27 M29
N28 G145 Y35 E10 Y7=2 F2=500
N29 G0 Y50
N30 M28
N31 G29 E11=E10=0 E11 N=41
N32 E5=E1-E2
  
```

N33 E6=(E5-E3):2
 N34 G29 E20=E5>E15 E20 N=43
 N35 G29 E20=E5>E16 E20 N=45
 N36 G149 T=E30 R1=4
 N37 G150 T=E30 R1=E4+E6
 N38 S1000 T1 M6 (MILL Diameter 18 mm)
 N39 G0 X140 Y50 Z-10 B0 F400 M3
 N40 G29 E20 E20=1 N=43
 N41 M0
 N42 (PROBE NOT TRIGGERED, NO MEASUREMENT EXECUTED)
 N43 G29 E20 E20=1 N=43
 N44 M0
 N45 (GROOVE WIDTH TOO BIG)
 N46 M30

Explanation

N1 : Define the plane of operation
 N2 : Activate a stored zero offset
 N3 : Set the maximum width of the groove
 N4 : Set the minimum width of the groove
 N5 : Calculate the width of the groove with the average tolerance
 N6 : The mill of 18 mm diameter is loaded as tool 1
 N7 : Start the spindle and move tool to start position
 N8 : Mill through the centre of the groove
 N9-N14: Mill the sides of the groove
 N15 : Cancel radius compensation
 N16 : Move the tool to the middle of the groove
 N17 : Retract the tool and stop the spindle
 N18 : Pick up the number of the actual tool
 N19 : Load the touch trigger probe
 N20 : Set the probe in an oriented position. It depends on the machine tool if this setting is necessary.
 N21 : Activate the probe
 N22 : Move the probe to the middle of the groove and at depth
 N23 : Activate the air pressure. The code of the M-function depends on the machine tool
 N24 : Measure the upper side of the groove. Store the measured value in the Y-axis at E1.
 Store the status of the measuring probe at E10
 N25 : Move the tool back to the middle of the groove
 N26 : Check to see if a measurement was executed E10=0 no measurement. Jump to N29 to switch off the probe and to display an error message.
 N27 : Activate the air pressure.
 N28 : Measure the lower side of the groove. Store the measured value in the Y-axis at E2
 Store the status of the measuring probe at E10
 N29 : Move the tool back to the middle of the groove
 N30 : Switch off the probe
 N31 : Check to see if a measurement was executed E10=0 No measurement. Jump to N41 to display an error message.
 N32 : Calculate the actual width of the groove from the measured Y-positions
 N33 : Calculate the difference between the programmed width and the measured width.
 The difference is related to the tool radius.
 N34 : Check to see if the measured tool width (E5) is greater than the maximum allowed width (E15). If greater jump to N43 to display an error.
 N35 : Maximum width is not exceeded. Check to see if the measured width (E5) is greater than the minimum allowed width. If greater the groove is finished. So jump to the end of the program (N44).

- N36 : The measured width is less than the minimum value. In this case the tool radius is changed in the tool memory and the sides of the groove milled again with the new radius value from the tool memory. Read the tool radius from the tool memory and store its value in parameter E4.
- N37 : Store the recalculated tool radius in the tool memory.
- N38 : The mill is loaded again
- N39 : Start the spindle and move tool to start position
- N40 : If the milling is finished, jump to the end of the program.
- N41 : Program stop
- N42 : Displaying an error text
- N43 : If the program continues, a jump to the end of the program is executed
- N44 : Program stop
- N45 : Displaying an error text
- N46 : End of program

EXAMPLE 4: Alignment of workpiece mounted on a rotary table

Only two points in X or Y have to be measured to be able to adjust a workpiece mounted on the rotary table rotating around the Z axis.

The angle between workpiece and X axis is calculated automatically by the CNC and may be used to rotate the table for positioning the workpiece parallel to the X axis.

If the workpiece is inclined towards the X axis at the beginning, this angle may be programmed with the C word.

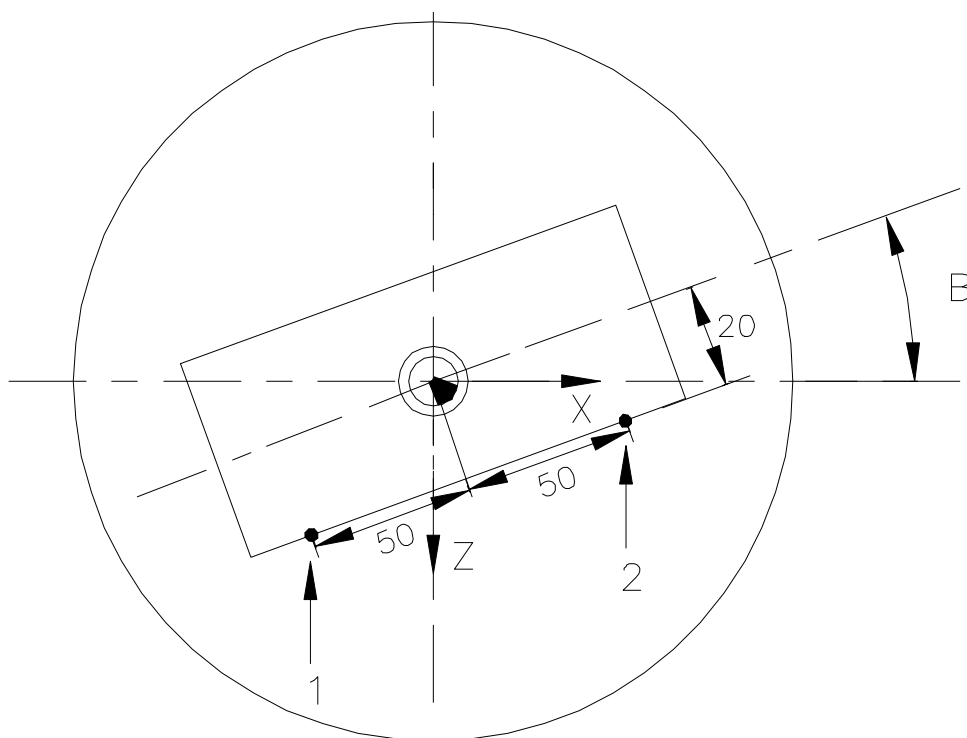
If C has not been programmed, C0 is used automatically instead of C.

Note:

This function can only be used when:

1. the rotary table is located in the XY plane, the workpiece is rotated around the Z axis (C axis) and the measuring probe is in Z direction.
2. the measurements are carried out in Y direction.

A part is mounted on a rotary table and should be aligned parallel to the X-axis. With a touch trigger probe two points on the part are measured and then the table is rotated over the calculated angle.



```

N50003
N1 G17
N2 G54
N3 T1 M6
N4 G0 X-50 Y-30 Z100 C0
N5 G1 Z0
N6 M27
N7 G145 X-50 Y-20 X7=11 Y7=12
N8 G0 Y-30
N9 G0 X50
N10 G145 X50 Y-20 X7=21 Y7=22
N11 G0 Y-30
N12 M28
N13 G0 Z100
N14 E30=(E12-E22)
N15 E31=(E21-E11)
N16 E32=(E30:E31)
N17 E33=atan(E32)
N18 G150 C7=E33 N1=54
N19 G54
N20 G0 C0
N21 M30

```

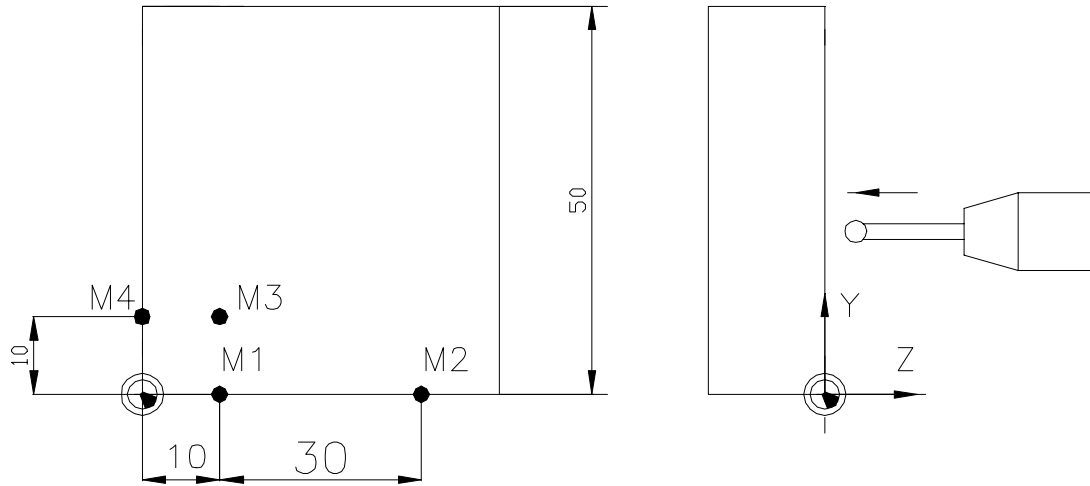
Explanation:

```

N1 : Set the plane of operation
N2 : Set the zero point
N3 : Load the touch trigger probe
N4 : Positioning first measuring point
N5 : Going to depth
N6 : Activate probe
N7 : Measure point 1 in direction Y (X in E11, Y in E12)
N8 : Retract
N9 : Positioning second measuring point
N10 : Measure point 2 in direction Y (X in E21, Y in E22)
N11 : Retract
N12 : Deactivate probe
N13 : Retract Tool
N14 : Calculate the difference (E30) in Y direction
N15 : Calculate the difference (E31) in X direction
N16 : Calculate Quotient (E32)
N17 : Calculate ARCTAN (E33)
N18 : A angle correction is made in the zero point of the C-Axis
N13 : Retract Tool
N14 : Calculate the difference (E30) in Y direction
N15 : Calculate the difference (E31) in X direction
N16 : Calculate Quotient (E32)
N17 : Calculate ARCTAN (E33)
N18 : A angle correction is made in the zero point of the C-Axis
N19 : Set the zero point
N20 : Rotate table back to C0
N21 : Program end

```

EXAMPLE 5 : Determining the zero point



The probe is standing in the Z-axis. The part is mounted on a table rotating around the Z-axis. Five points of the part (M1 to M5) are measured. M1 and M2 for covering the angular displacement; M3, M3 and M5 for measuring the positions of the axes.

The section of the part program for determination the zero point could be:

```

N50004
N1 G54
N2 G17
N3 T1 M6 (probe)
N4 G0 X10 Y-10 Z70 C0 F1000
N5 G1 Z-5
N6 M27
N7 G145 X10 Y0 X7=11 Y7=12
N8 G0 Y-10
N9 G0 X50
N10 G145 X50 Y0 X7=21 Y7=22
N11 G0 Y-10
N12 G0 Z70
N13 E30=(E12-E22)
N14 E31=(E21-E11)
N15 E32=(E30:E31)
N16 E33=atan(E32)
N17 G150 C7=E33 N1=54
N18 G54
N19 G0 C0
N20 G0 X10 Y10 Z10
N21 G145 X10 Y10 Z0 Z7=3
N22 G0 Z10
N23 G0 X-10 Y10
N24 G1 Z-5

```



```

N25 G145 X0 Y10 Z-5 X7=1
N26 G0 X-10
N27 G0 Z10
N28 G0 X10 Y-10
N29 G1 Z-5
N30 G145 X10 Y0 Z-5 Y7=2
N31 G0 Y-10
N32 G0 Z50
N33 G150 X7=E1 Y7=E2 Z7=E3 N1=54
N34 G54
N35 M28
:

```

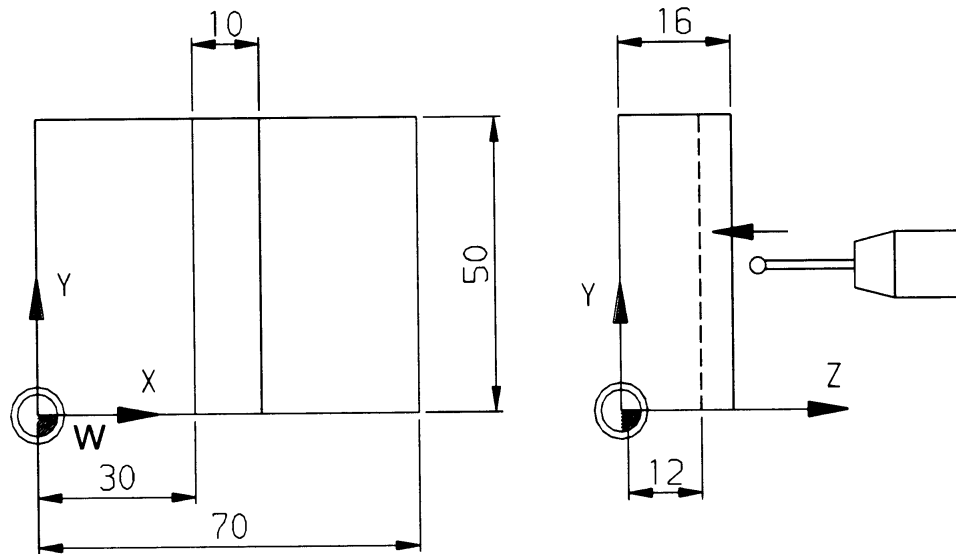
Explanation:

```

N1 : Set the zero point
N2 : Set the plane of operation to be XZ-plane
N3 : Load the touch trigger probe
N4 : Move to programmed position
N5 : move to depth in hole 1
N6 : Activate probe
N7 : Measure point M1 (X in E11, Y in E12)
N8 : Retract probe to avoid collision
N9 : Move to position for measure point 2
N10 : Measure point M2 (X in E21, Y in E22)
N11 : Retract probe to avoid collision
N12 : Retract probe to avoid collision
N13 : Calculate the difference (E30) in Y direction
N14 : Calculate the difference (E31) in X direction
N15 : Calculate quotient (E32)
N16 : Calculate ARCTAN (E33)
N17 : A angle correction is made in the zero point of the C-Axis
N18 : Set the zero point
N19 : Rotate table back to C0
N20 : Move to measure point 3
N21 : Measure point M3, to determine the position in the Tool axis(Z in E3)
N22 : Retract probe to avoid collision
N23 : Move to measure point 4
N24 : Move to depth
N25 : Measure point M4, to determine the position in the X axis(X in E3)
N26 : Retract probe to avoid collision
N27 : Move to measure point 1
N28 : Move to depth
N29 : Measure point M1, to determine the position in the Y axis(Y in E3)
N30 : Retract probe to avoid collision
N31 : Update of the zero offset values in the X-, Y- and Z-Axis
N32 : Set the updated zero point
N33 : Deactivate probe

```

EXAMPLE 6: Correcting the length of a tool



```

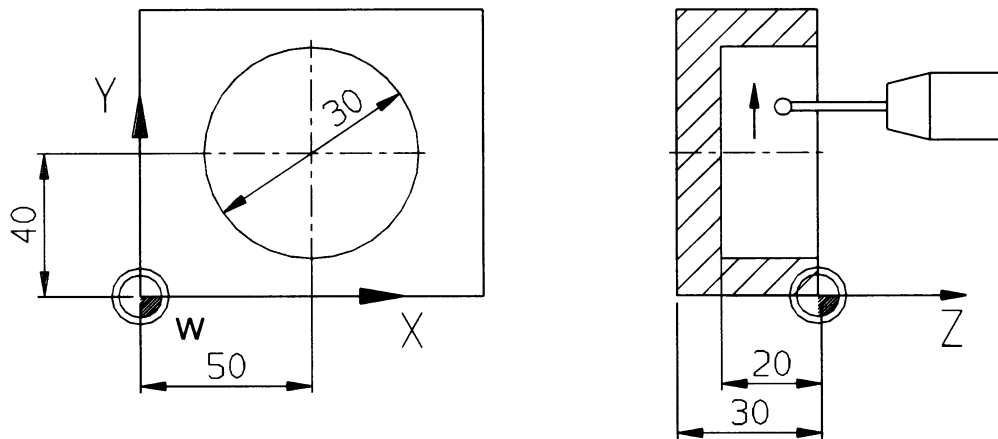
N90005
N1 G17
N2 T1 M6 (Mill R5)
N3 G0 X35 Y60 Z12 S1000 M3
N4 G1 Y-10 F200
N5 G0 Z200 M5
N6 T2 M6 (Prober)
N7 G0 X35 Y25 Z20
N8 M27
N9 G145 X35 Y25 Z12 Z7=1
N10 G149 T1 L1=2
N11 G150 T1 L1=E2-E1+12
N12 M28
N13 Z200 M30

```

Explanation:

N1 : Set the plane of operation to be the XY-plane
 N2 : Load the mill of 10 mm diameter
 N3 : Start the spindle and move the mill to the start point
 N4 : Mill the groove
 N5 : Retract the tool and stop the spindle
 N6 : Load the probe
 N7 : Move to start point
 N8 : Activate probe
 N9 : Measure the tool axis in negative direction (Z in E1)
 N10 : Store the tool length T1 in E2
 N11 : The tool length is dependent of the calculated difference in the Z-Axis.
 N12 : Deactivate probe
 N13 : Retract probe and end program.

EXAMPLE 6 : Milling and measuring a hole



A hole is milled and measured with a touch trigger probe. Checks are provided to see if the tolerance on the radius of the hole is within the required limits. If the radius is too small, the hole is milled again. If the radius is too large the part is rejected and an message displayed.

The part program could be:

```

N50006
N1 G54
N2 G17
N3 T1 M6 (mill R=5)
N4 E1=50 (X drilling centre point)
N5 E2=40 (Y drilling centre point)
N6 E3=0 (Z drilling start point)
N7 E4=20 (hole depth)
N8 E5=15 (drill radius)
N9 E6=3 (probe radius)
N10 (E7 Measure status)

N11 G89 Z=-E4 B2 R=E5 K6 F300 S1000 M3
N12 G79 X=E1 Y=E2 Z=E3
N13 G0 Z50 M5

N14 T2 M6 (Probe)
N15 M27
N16 G0 X=E1+E5-E6-2 Y=E2 Z50
N17 G1 Z-5
N18 G145 X=E1+E5-E6 Y=E2 L0 E7
N19 G29 E0 E0=E7<1 N=49
N20 G145 X=E1+E5-E6-2 Y=E2 L1 X7=10
N21 G0 X=E1+E5-E6-2 Y=E2
N22 G3 I=E1 J=E2 X=E1 Y=E2+E5-E6-2
N23 G145 X=E1 Y=E2+E5-E6 L0 E7
N24 G29 E0 E0=E7<1 N=49
N25 G145 X=E1 Y=E2+E5-E6-2 L1 Y7=11
N26 G0 X=E1 Y=E2+E5-E6-2
N27 G3 I=E1 J=E2 X=E1-E5+E6+2 Y=E2
N28 G145 X=E1-E5+E6 Y=E2 L0 E7
N29 G29 E0 E0=E7<1 N=49
  
```

N30 G145 X=E1-E5+E6+2 Y=E2 L1 X7=12
 N31 G0 X=E1-E5+E6+2 Y=E2
 N32 G3 I=E1 J=E2 X=E1 Y=E2-E5+E6+2
 N33 G145 X=E1 Y=E2-E5+E6 L0 E7
 N34 G29 E0 E0=E7<1 N=49
 N35 G145 X=E1 Y=E2-E5+E6+2 L1 Y7=13
 N36 G0 X=E1 Y=E2-E5+E6+2
 N37 G0 Z50
 N38 M28

N39 E20=E1+(E10+E12):2 (measured X coordinate)
 N40 E21=E2+(E11+E12):2 (measured Y coordinate)
 N41 E22=sqrt((E10-E20)^2+(E2-E21)^2) (First radius hole)
 N42 E22=E22+sqrt((E11-E21)^2+(E1-E20)^2) (Second radius hole)
 N43 E22=E22+sqrt((E12-E20)^2+(E2-E21)^2) (Third radius hole)
 N44 E22=(E22+sqrt((E13-E21)^2+(E1-E20)^2)):4 (Radius hole)
 N45 G29 E1 E1=1 N=51

N46 M0
 N47 (HOLE OUT OF TOLERANCE)
 N48 G29 E0 E0=1 N=51
 N49 M0
 N50 (NO TRIGGER POINT FOUND)
 N51 M30

Explanation:

N1 : Define the plane of operation
 N2 : Activate a stored aero offset
 N3 : Load mill (d=20 mm)
 N4-N10 : Edit e-parameters
 N11 : Define drill cycle
 N12 : Drill hole
 N13 : Retract tool and stop spindle
 N14 : Load probe
 N15 : Activate probe
 N16 : Move to the first hole
 N17 : Move to depth
 N18 : Measure first position with high feedrate
 N19 : Measuring position reached, but no measurement was carried out. Jump to error message (E7=0)
 N20 : Measure first measuring position with measurement feed (E10 is measuring position in X direction)
 N21 : Return to starting position (first measuring position)
 N22 : Move to the second hole with a circular movement
 N23 : Measure second position with high feedrate
 N24 : Measuring position reached, but no measurement was carried out. Jump to error message (E7=0)
 N25 : Measure second measuring position with measurement feed (E11 is measuring position in Y direction)
 N26 : Return to starting position (first measuring position)
 N27 : Move to the third hole with a circular movement
 N28 : Measure third position with high feedrate
 N29 : Measuring position reached, but no measurement was carried out. Jump to error message (E7=0)
 N30 : Measure third measuring position with measurement feed (E12 is measuring position in X direction)

N31 : Return to starting position (first measuring position)
N32 : Move to the third hole with a circular movement
N33 : Measure fourth position with high feedrate
N34 : Measuring position reached, but no measurement was carried out. Jump to error message (E7=0)
N35 : Measure fourth measuring position with measurement feed (E13 is measuring position in Y direction)
N36 : Return to starting position (first measuring position)
N37 : Retract probe to avoid collision
N38 : Deactivate probe
N39 : Calculate X centre point (E20)
N40 : Calculate Y centre point (E21)
N41 : Calculate first radius (Distance between measure position and the new centre point)
N42 : Add second radius
N43 : Add third radius
N44 : Add fourth radius and divide by four.
N45 : Jump to the end of the program
N46 : Stop program
N47 : Error message
N48 : Jump to the end of the program
N49 : Stop program
N50 : Error message
N51 : Program end

5.56 G148 Reading measure probe status

Purpose

To read the probe status in measuring cycle macros.

Format

N... G148 {I1=...} E...

Parameter

G	Reading measure probe status
E	E-parameter for probe status
I1=	Status group (1-3)

Type of function

Non-modal

Associated Functions

G145, G149, G150

Notes and usage

PROBE STATUS

I1=1 or not programmed.

The E-parameter can have one of four values:

- | | |
|-----------|--|
| E... = 0: | The probe is not activated; no contact with measuring surface. The E-parameter with the measured value is unchanged. |
| E... = 1: | The probe is activated; in contact with the measuring surface. The measured value is stored in the E-parameter |
| E... = 2: | Block Search, Testrun or the Demo mode is active. |
| E... = 3: | A probe error is active; no measurements can be made. |

The priority for probe status codes is:

Priority:

- | | |
|----|-----------------------------|
| 1: | code 2 (active mode) |
| 2: | code 3 (probe error) |
| 3: | code 0 or 1 (probe contact) |

I1=2

- | | |
|-----------|---|
| E... = 0: | during the measurement no measuring point is determinate. |
| E... = 1: | during the measurement a measuring point is determinate. |

I1=3

E... = 0: information of IPLC: probe/laser not connect
E... = 1: information of IPLC: probe/laser connect
See documentation measuring system.

INTERRUPT

The G148 function cannot be stopped by an interrupt command.

Example

N110 G148 E27
N115 G29 E91=E27=2 E91 N=300

Explanation:

N110: Store probe status in E-parameter number 27.
N115: Jump to block N300 if the program is executed in Block Search, Testrun or Demo. In this way e.g. calculation with parameters, which are not loaded while no measurement was executed, can be avoided.

Note: The function G148 is not permitted when G182 is active.

5.57 G149 Reading tool data or zero offset values

Purpose

To read tool data or zero offset values and store them in specified E-parameters within measuring cycle macros.

Format: Tool data

To read active tool number:

N... G149 T0 E...

To read tool dimensions:

N... G149 T... {T2=...} {L1=...} {R1=...} {M1=...}

To read tool status:

N... G149 T... E...

Format: Zero offsets

To read active zero offset G-functions:

N... G149 N1 =0/1 E...

To read stored pallet offsets:

N... G149 N1=52 [(axis address)7=...] {(axis address)7=...}

To read stored zero offsets:

With standard zero offsets or MC84=0:

N... G149 N1=54...59 [(axis address)7=...] {(axis address)7=...}

With MC84>0 zero offsets extends:

N... G149 N1=54.[nr] [(axis address)7=...] {(axis address)7=...}{B47=...}

In G54.[nr] the number [nr] must be given in two digits (G54.01 and G54.10).

To read programmable zero offset:

N... G149 N1 =93 [(axis address)7=...] {(axis address)7=...}

Format: Actual position

To read the actual position of X, Y or Z:

N... G149 [(axis address)7=...] {(axis address)7=...}

Parameter

G	Reading tool or offset values
T	Tool number
E	E-parameter
N1=	Zero offset shift
B47=	E-parameter for rotation in B4=
X7=	E-par. for offset/position in X
Y7=	E-par. for offset/position in Y
Z7=	E-par. for offset/position in Z
A7=	E-par. for offset/position in A
B7=	E-par. for offset/position in B
L1=	E-parameter for tool length
R1=	E-parameter for tool radius
T2=	Tool offset index
M1=	E-parameter for tool life

Tool data

T	Tool number
T2=	Tool offset index
E	E-parameter
L1=	E-parameter for toollength
R1=	E-parameter for tool radius
M1=	E-parameter for toollife

Zero offsets

N1=	Zero offset shift
X7=	E-parameter for offset /position in X
Y7=	E-parameter for offset /position in Y
Z7=	E-parameter for offset /position in Z
A7=	E-parameter for offset /position in A
B7=	E-parameter for offset /position in B
C7=	E-parameter for offset /position in C
B47=	E-parameter for rotation in B4=

Type of function

Non-modal.

Associated Functions

G145, G148, G150

Notes and usage**TOOL NUMBER (T)**

Number of tool for which tool data must be read. If the FMS-tool memory (Flexible Manufacturing System) is in use, the complete number including the spare tool index has to be written.

TOOL DATA

The tool radius (R1=..), toollength (L1=..) and the remaining toollife (M1=..) can be read.

TOOL OFFSET INDEX (T2=)

A tool-offset index 0,1 or 2 can be specified. Default is T2=0.

When T2=0

Tool radius = radius (R) + radius oversize (R4=)

Toollength = length (L) + length oversize (L4=)

Better is to use G321.

TOOL STATUS (E)

The tool status from the tool memory will be loaded in the indicated E-parameter. The tool status can have the following values.

E... = 1: the tool is enabled and measured.

E... = 0: the tool is enabled but not measured

E... = -1: the tool is disabled

E... = -2: the tool time is expired

E... = -4: tool breakage error

E... = -8: the tool cutting force is exceeded

E... = -16: the tool time < T3 = programmed

A combination of error messages is also possible:

E... = -13 means: error message -8 and -4 and -2 and 1.

ZERO OFFSET NUMBER (N1=)

The number of the zero offset which data has to be read. 'N1=' can have a value from 51 to 59, 54.[nr] or 92/93. G92 gives the same result as G93 (absolute).

ZERO OFFSET GROUP (N1=)

The zero offset group of which the active G-function has to be read.

N1= can have the value 0 or 1.

N1=0: If G52 is active, the E-parameter is given the value 52. If G52 is not active, the E-parameter is given the value 51.

N1=1: The E-parameter is given the value of the active offset G54 - G59. IF a G54-G59 type offset is not active, the E-parameter is given the value 53.

READING ACTUEL AXES POSITION VALUES (X7, Y7, Z7)

The Axes position values can be read out in E-Parameter. X7=20 means: E20 is filled with the actual axes position values. (See also G326).

READING ADDRESSES WITHOUT VALUE:

If addresses are read from the tool memory when they are not entered previously, a value of zero will be returned.

INTERRUPT

The function cannot be stopped by an interrupt command.

Note: The function G149 is not permitted when G182 is active.

The tool data of T0 cannot be read. If T0 is used, the relevant E-parameters are not loaded. No error message is given to this effect.

Examples

EXAMPLE 1: to read the active tool number.

N100 G149 T0 E1

E1 contains the number of the active tool

EXAMPLE 2: to read the active tool dimensions.

N100 G149 T=E1 L1=5 R1=6

Read the tool dimensions of tool T=E1

E5 contains the tool length (E5 = length (L) + length oversize (L4=))

E6 contains the tool radius (E6 = radius (R) + radius oversize (R4=))

Better is to use G321.

EXAMPLE 3: to read the active tool dimensions.

N100 G149 T12 L1=5 R1=6

Read the tool dimensions of tool T12

E5 contains the tool length (E5 = length + length oversize (L4=))

E6 contains the tool radius (E6 = radius (R) + radius oversize (R4=))

Better is to use G321.

EXAMPLE 4: to read the active zero offset function

N100 G149 N1=0 E2

N110 G149 N1=1 E3

E2 contains the active preset function (51 or 52)

E3 contains the active zero offset function (53 to 59) or G54.[nr]

EXAMPLE 5: to read a stored zero offset.

N100 G149 N1=54 X7=1 Z7=2

or

N100 G149 N1=54.[nr] X7=1 Z7=2

Read offset G54.

E1 contains X-axis offset.

E2 contains Z-axis offset.

EXAMPLE 6: Calling a shift with angle of rotation of coordinate system

N100 G149 N1=54.02 X7=1 B47=2

Call shift G54.02

E1 has shift in X

E2 has angle of rotation of coordinate system

Example 7: Calling a zero point shift (G92/G93)

N100 G149 N1=92 X7=1 Z7=2

Call shift G92

E1 has shift in X

E2 has shift in Z

If a G92/G93 shift is called while G92/G93 is ineffective, the shift values 0 are obtained.

5.58 G150 Change tool data or zero offset values

Purpose

To write values in the tool memory or zero offset memory within measuring cycle macros.

Format: Tool data

To write data in tool memory

N... G150 T... {T2=...} {L1=...} {R1=...} {M1=...}

**When T2=0, oversize (L4= or R4=) set to zero.
Better is to use G331.**

To write tool status in tool memory:

N... G150 T... E...

Format: Zero offsets

To write data in zero offset memory:

With standard zero offsets or MC84=0:

N... G150 N1=54...59 [(axis address)7=...] {(axis address)7=...} etc.

With MC84>0 zero offsets extends:

N... G150 N1=54.[nr] [(axis address)7=...] {(axis address)7=...} etc.

In G54.[nr] the number [nr] must be given in two digits (G54.01 and G54.10).

Parameter

```
G   Change tool or offset values
T   Tool number
E   E-parameter
N1= Zero offset shift
B47= Angle of rotation in B4=
X7=  Offset in X
Y7=  Offset in Y
Z7=  Offset in Z
A7=  Offset in A
B7=  Offset in B
L1=  Toollength value in T
R1=  Tool radius value in T
T2=  Tool offset index
M1=  Tool life value in T
```

Tool data

T	Tool number
T2=	tool offset index
E	E-parameter
L1=	Toollength value in T
R1=	Tool radius value in T
M1=	Toollife value in T

Zero offsets

N1=	Zero offset shift
X7=	Offset in X
Y7=	Offset in Y
Z7=	Offset in Z
A7=	Offset in A
B7=	Offset in B
C7=	Offset in C
B47=	Angle of rotation in B4=

Type of function

Non-modal

Associated Functions

G145, G148, G149

Notes and usage

TOOL NUMBER (T)

Number of tool for which tool data must be changed. If the FMS-tool memory (Flexible Manufacturing System) is in use, the complete number including the spare tool index has to be written.

The modal number of the actual tool is not influenced by this command.

TOOL OFFSET INDEX (T2=)

A tool-offset index 0,1 or 2 can be specified. Default is T2=0. The offset index of the actual tool is not influenced by this command.

TOOL STATUS (E)

The tool status can be loaded from the indicated E-parameter into the tool memory.

Possible values for tool status are:

E... = 1: the tool is enabled and measured.

E... = 0: the tool is enabled but not measured

E... = -1: the tool is disabled

E... = -2: the tool time is expired

E... = -4: tool breakage error

E... = -8: the tool cutting force is exceeded

E... = -16: the tool time < T3 = programmed

A combination of error messages is also possible:

E... = -13 means: error message -8 and -4 and -2 and 1.

ZERO OFFSET (N1=)

Zero offset (G52, G54-G59 or **G54.[nr]**) that has to be changed.

INTERRUPT

The function cannot be stopped by an interrupt command.

Note: The function G150 is not permitted when G182 is active.
The tool data of T0 cannot be loaded.

Examples

EXAMPLE 1. Write data in the tool memory.

N50 G150 T1 L1=E2 R1=4 M1=10

Explanation:

Change the data of tool No.1.
Store the value of parameter E2 as the tool length.
Make the tool radius equal to 4.
Make rest tool lifetime equal to 10 minutes.

When T2=0, the oversize (L4= or R4=) is set to zero.
Better is to use G331.

N50 G331 T1 I1 E2 (writing length)
N50 G331 T1 I4 E.. (writing length oversize)

EXAMPLE 2. Write data in the zero offset memory.

N70 G150 N1=57 X7=E1 Z7=E6
or
N70 G150 N1=54.3 X7=E1 Z7=E6

Explanation:

Zero offset values of G57 are to be changed.
Store value of parameter E1 in G57 or G54.3 X-axis offset
Store value of parameter E6 in G57 or G54.3 Z-axis offset

EXAMPLE 3: Changing a zero point shift with angle of rotation of coordinate system:

N70 G150 N1=54.03 X7=E1 B47=E6

Explanation:

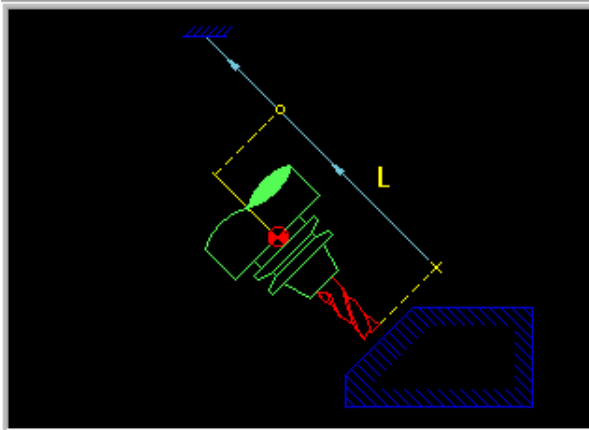
Change zero point shift values of G54.03
Store value of E1 parameter in G54.3 shift in X
Store value of E6 parameter in G54.3 shift in B4=

5.59 G174 Tool withdrawal movement

Purpose

Movement to move the tool axis clear during 5-axis milling.

With this function, you are always able to move away in the direction of the tool axis (only if Z axis is programmed). The tool is withdrawn until the 'first' SW limit switch is reached. The direction of movement is determined by the position of the milling head. The calculation is carried out with the help of the kinematic model.



Format

N.. G174 {L....}

Parameter

G	Tool retract movement
L	Retract distance

Modal word
F6=

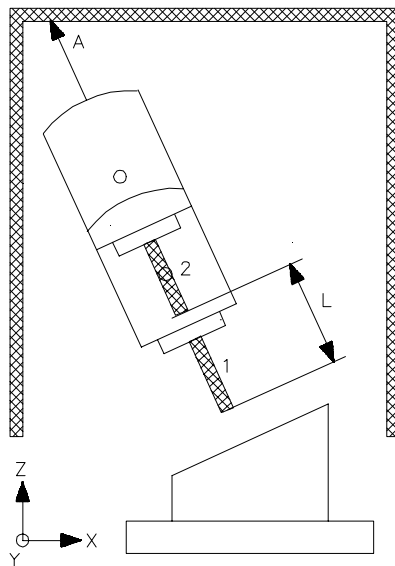
Type of function

Non-modal

Associated function**NOT PERMITTED G-FUNCTIONS, WHEN ACTIVATING G174**

Before G174 is programmed, the next (modal) G-functions must be switch off:
G64, G197, G198, G199, G200, G201, G203, G204, G205, G206, G207, G208

When one of these G-functions is active, the error message P77 'G-function and Gxxx not permitted' is given.

Notes and usage

- 1** Starting position
- L** Withdrawal distance
- 2** End position
- A** Limitation by software limit switch

WITHDRAWAL DISTANCE (L)

The withdrawal distance defines the distance travelled in the direction of the tool.

L is always positive. The maximum travel distance is limited by the software limit switch.

The basic setting is with the software limit switch.

DIRECTION OF MOVEMENT

The direction of movement is determined by turning the milling head. This is calculated with the help of the kinematic model.

The function is always active.

EXECUTION (G0 or G1)

G174 is executed in rapid motion or if F6= <Feed per block> is programmed with F6=.

Following G107, G0 or G1 from the previous block is modally active again.

Example Tool withdrawal movement.

N10 G174 L100 Tool retracts 100 mm or till the software limit switches.

5.60 G180 Basic coordinate system

Purpose

This function has two meanings:

- 1) Deactivating cylindrical coordinates systems (G182).
- 2) Defining of mainplane and tool axis (Basic coordinates system).

Format

G180: Basic coordinate system

N... G180 [auxiliary axis 1][auxiliary axis 2][Tool axis]

Parameter

```
G   Cancel cylinder interpolation
X   X1:allocate axis to coord.system
Y   Y1: allocate axis to coord.system
Z   Z1: allocate axis to coord.system
A   A1:allocate axis to coord.system
B   B1:allocate axis to coord.system
```

General principles

The normal expression is G180 X1 Y1 Z1

The only following configurations are possible:

Auxiliary axis 1	X
Auxiliary axis 2	Y
Tool axis	Z or W

The correct procedure depends on 3 different types of information:

- 1) The tool axis is determined by G17/G18/G19 (G17 Z).
- 2) G180 determines which axes are to be substituted. (G17 W in Z)
- 3) The machine constants for the tool axis definition should also be correct (tool axis W belongs to Z).

Associated functions

G182

Type of function

Modal

Notes and usage**FUNCTIONS TO BE CANCELLED**

The functions G41-G44, G64, G73, axis rotation (G92/G93 B4=) and G141 must be cancelled before G180 is activated.

Any other function, which is active immediately before the G180 block, remains active.

Note: The words X, Y, Z cannot be programmed without any value. Therefore the value 1 is written to it. This value has no meaning.

RADIUS AND TOOL LENGTH COMPENSATION

The tool length compensation is active in the defined tool axis. The radius compensation is active in the auxiliary axis.

MACHINE CONSTANT

The machine constant must be accurate. If W-Axis is the fourth axis then MC117 = 3 (just like Z-Axis). MC3401 = 0 (W-Axis is a Linear axis).

COORDINATES

Only Cartesian coordinates can be used.

Note: If G180 is programmed and radius compensation is still active, the compensation is cancelled with the G180. It is advised to cancel radius compensation with G40 and then to return to the basic coordinate system.

CANCELLATION

Cylinder interpolation is cancelled by either the G180 function or CLEAR CONTROL.

DEFAULT MODE

When turning on the controller or activating Softkey CLEAR CONTROL G180 X1 Y1 Z1 is automatically turned on.

Example

```
N12340
N1 G17 S1000 T1 M6
N2 G54
N3 G180 X1 Y1 W1
N4 G81 Y2 B10 Z-22 F1000 M3
N5 G79 X0 Y0 Z0
```

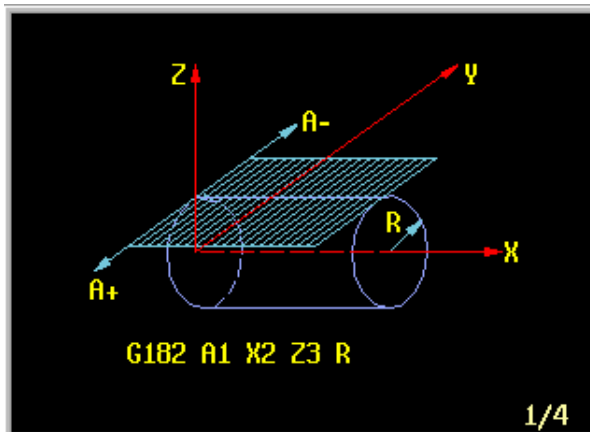
Explanation:

N1: Define plane of operation.
 N2: Activate a stored zero offset.
 N3: Activate auxiliary plane XY and Tool axis W.
 N4: Define drilling cycle.
 N5: Drilling hole with the feedrate active in the W axis.

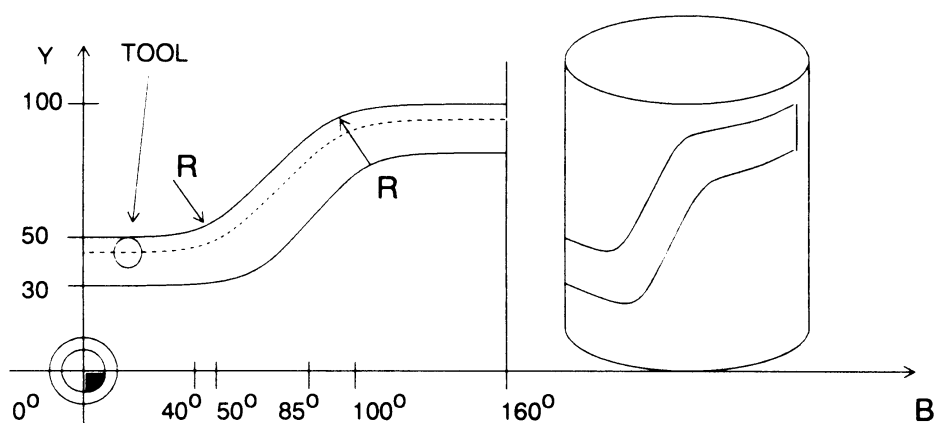
5.61 G182 Cylindrical coordinate system

Purpose

Selection of the cylindrical coordinate system. Using this system, you can easily program contours and positions on the curved surface of a cylinder.



Format



NB8696

G182: Cylindrical coordinate system

To activate the cylindrical coordinate system

N.. G182 [cylinder axis] [rotary axis] [rotary axis] R..

N.. G182 R..

Format with active G182

Rapid

N... G0 [cylinder axis] [rotary axis] {rotary axis}

Linear movement:

N... G1 [cylinder axis] [rotary axis] {rotary axis} {F..}

Circular movement:

N... G2/G3 [cylinder axis] [rotary axis] R..

Parameter

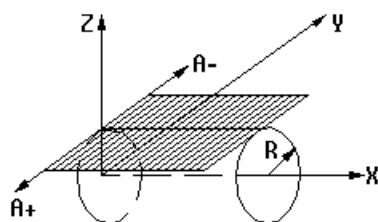
```

G   Activate cylinder interpolation
X   Cylinder plane:2 / Tool axis:3
Y   Cylinder plane:2 / Tool axis:3
Z   Cylinder plane:2 / Tool axis:3
A   Cylinder plane:1
B   Cylinder plane:1
R   Cylinder radius

```

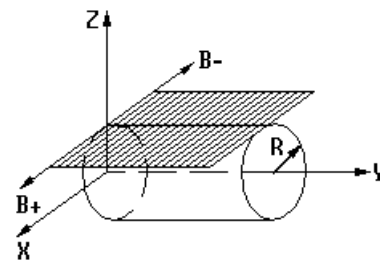
Modal words:

S, T, H, M, E, F1=, T1=, T2=



G182 A1 X2 Z3 R

1/4

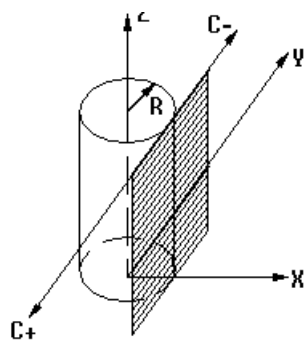


G182 B1 Y2 Z3 R

2/4

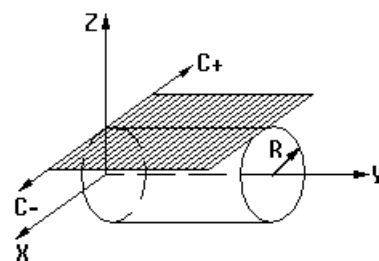
G182 A1 X2 Z3 R..
or (as previous)
G182 A1 X1 Z1 R..

G182 B1 Y2 Z3 R..
or (as previous)
G182 B1 Y1 Z1 R..



G182 C1 Z2 X3 R

3/4



G182 C1 Y2 Z3 R

4/4

G182 C1 Z2 X3 R..
or (as previous)
G182 C1 X1 Z1 R..

G182 C1 Y2 Z3 R..

Associated functions

G180

Type of function

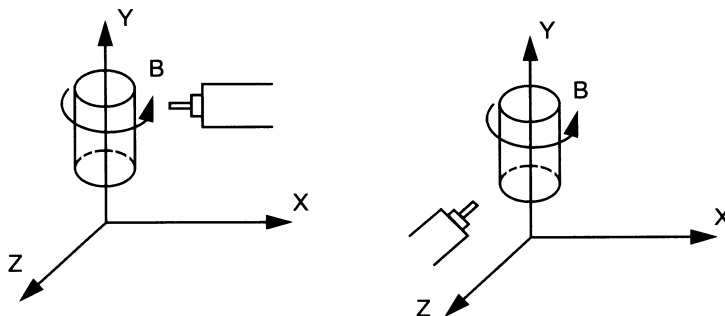
Modal

General principles

Contours on the curved surface of a cylinder are drawn in a plane representing the curved surface. This plane is defined with the rotary axis, the cylinder axis and the cylinder radius. In this plane linear and circular movements with radius compensation can be programmed. During the execution of the program these movements are converted to movements with a linear (= the axis of the cylinder) and a rotary axis (= the axis rotated about the cylinder axis). This is called cylinder interpolation.

In general the following configurations are possible:

Rotary axes	A,	B,	C
Cylinder axes	X,	Y,	Z
Tool axis	Y or Z,	X or Z,	X or Y.



NB9661

The BY-plane for cylinder interpolation

The correct procedure depends on 3 different types of information:

- 1) The tool axis is determined by G17/G18/G19 (G17 Z).
- 2) G182 determines which axes are to be substituted. (G17 AX or BY)
- 3) The machine constants for the rotary axis definition should also be correct. (Rotary axis A belongs to X).

Notes and usage**FUNCTIONS TO BE CANCELLED**

The functions G41-G44, G64, G73, axis rotation (G92/G93 B4=) and G141 must be cancelled before G182 is activated.

Any other function, which is active immediately before the G182 block, remains active.

SPECIFYING THE CYLINDER PLANE

The words X, Y, Z, A, B, C cannot be programmed without any value.

In the G182-block the configuration for cylinder interpolation is programmed:

Standard configuration

Rotary axis:	A1	B1	C1
Cylinder axis:	X1	Y1	Z1
Tool axis:	Y1/Z1	X1/Z1	X1/Y1
Cylinder radius:	R	R	R

Other configuration

Rotary axis marked with 1:	A1	B1	C1
Cylinder axis marked with 2:	X2Y2Z2	Y2X2Z2	Z2X2Y2
Tool axis marked with 3:	Y3Z3X3	X3Z3Y3	X3Y3Z3
Cylinder radius:	R	R	R

Note: When the mark is 1 then only a standard configuration is possible.

MACHINE CONSTANTS

The machine constants must be accurate.

MC 102 = 1, MC103 = 88 (X-axis)

MC 107 = 2, MC108 = 89 (Y-axis)

MC 112 = 3, MC113 = 90 (Z-axis)

MC 117 = 4 belongs to axis 1 (4-3), MC118 = 65 (A-axis turns around X-Axis)

MC 122 = 6 belongs to axis 3 (6-3), MC123 = 67 (C-axis turns around Z-Axis)

DEFAULT CYLINDER PLANE

When a machine tool has only one rotary table, the configuration for cylinder interpolation is defined in the Machine Constants. Therefore, if the axis configuration is not programmed in a G182-block, these settings are used automatically by the CNC.

CYLINDER RADIUS

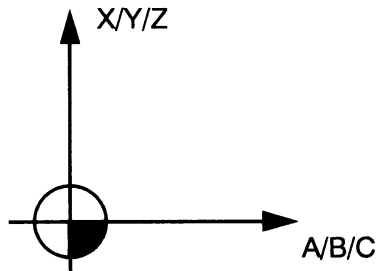
The cylinder radius is used by the CNC for calculating the feedrate of the rotary axis. The radius value must be between 1 mm and 500 mm. If the R-word is not programmed in a G182-block, an error message is displayed.

CHANGING THE CYLINDER RADIUS

The radius of the cylinder can be changed with the R-word in another G182-block. In this block the definition of the plane has to be repeated.

PLANE OF THE CURVED SURFACE

The plane of the curved surface is called either AX, BY or CZ, depending on which cylinder the contour has to be made.



NB9656

Axes of the plane of the curved surface

The horizontal axis is the rotary axis and is programmed with the corresponding axis address A, B, C in degrees and decimal parts thereof.

The vertical axis is the cylinder axis and is programmed with the corresponding axis address X, Y, Z in mm or inches.

The tool axis is perpendicular to the wall of the cylinder, programmed in mm or inches with the addresses Y or Z for the AX-plane, X or Z for the BY-plane, X or Y for the CZ-plane depending on which axis the tool is loaded.

DATUM POINT

The datum point in the rotary, cylinder and tool axis must be programmed, before cylinder interpolation is activated. This can be achieved with

G51-G52 (p) reset axes.
G53-G59 or G54I[nr.] stored zero offset
G92/G93 a datum point shift

Once cylinder interpolation (G182) is activated a datum point shift is not allowed until the basic coordinate system (G180) is chosen again.

COORDINATES

Only Cartesian coordinates can be used.

The functions G90 and G91 are used for programming absolute (G90) or incremental (G91) dimensions and can be used with the rotary, cylinder and tool axis.

RAPID MOVEMENTS (G0)

A rapid traverse movement is programmed with G0 and the end point of the movement. Two or three axes can be programmed in one block. The axes move with the positioning logic:

- | | | |
|----------------------------|----|-----------------------|
| Tool towards the cylinder: | 1. | movement in the plane |
| | 2. | tool axis movement |
| Tool from the cylinder: | 1. | tool axis movement |
| | 2. | movement in the plane |

LINEAR FEED MOVEMENTS (G1)

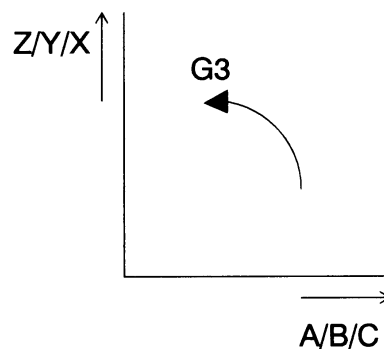
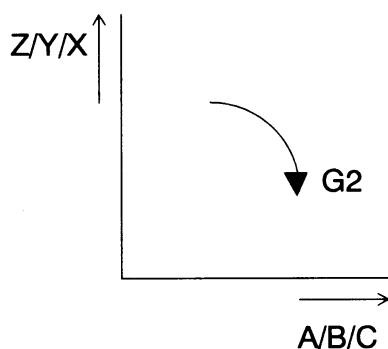
A linear feed movement is programmed with G1, the end point of the movement and the feedrate.

Two or three axes can be programmed in one block. All axes move simultaneously and reach their end point at the same time.

The programmed feed rate is the surface feed on the cylinder at the radius from the G182-block.

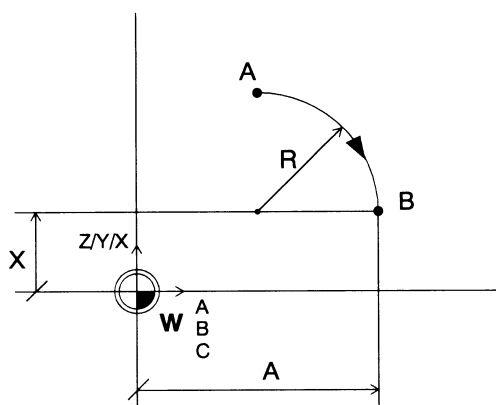
CIRCULAR FEED MOVEMENTS (G2/G3)

A circular feed movement can only be programmed with a G2 or G3, the end point coordinates and the radius of the arc (R-word).

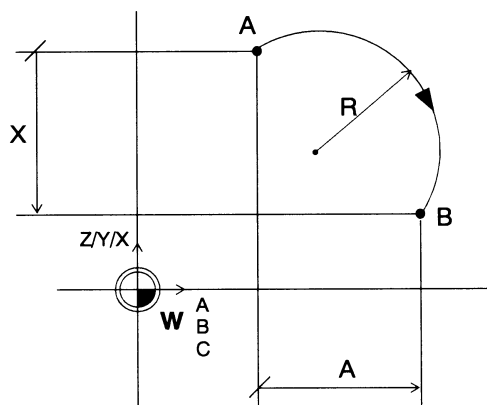


NB7970

Direction of circular movement



G90



G91

NB7971

Circular arc with end point and radius

RADIUS COMPENSATION

For radius compensation in the plane of the cylinder the functions G40, G41, G42, G43 and G44 can be used. These functions have the same meaning as in the basic coordinate system (G180 active). For defining LEFT and RIGHT one should look from the tool to the cylinder.

Note: If G180 is programmed and radius compensation is still active, the compensation is cancelled with the G180

It is advised to cancel radius compensation with G40 and then to return to the basic coordinate system.

TOOL SIZE

From the control side of view there is no restriction on the size of the tool radius. However, if the tool radius is too large, undercuts may be produced. These undercuts depend on the shape and size of the tool and the depth of operation.

Note: With contours on the cylinder the greatest accuracy is achieved with a tool of which the diameter is about 0.2 mm less than the width of the groove.

CANCELLATION

Cylinder interpolation is cancelled by either the G180 function or CLEAR CONTROL.

DEFAULT MODE

G180 is made active automatically when the CNC is switched on, or the CLEAR CONTROL operation is performed.

FUNCTIONS PERMITTED

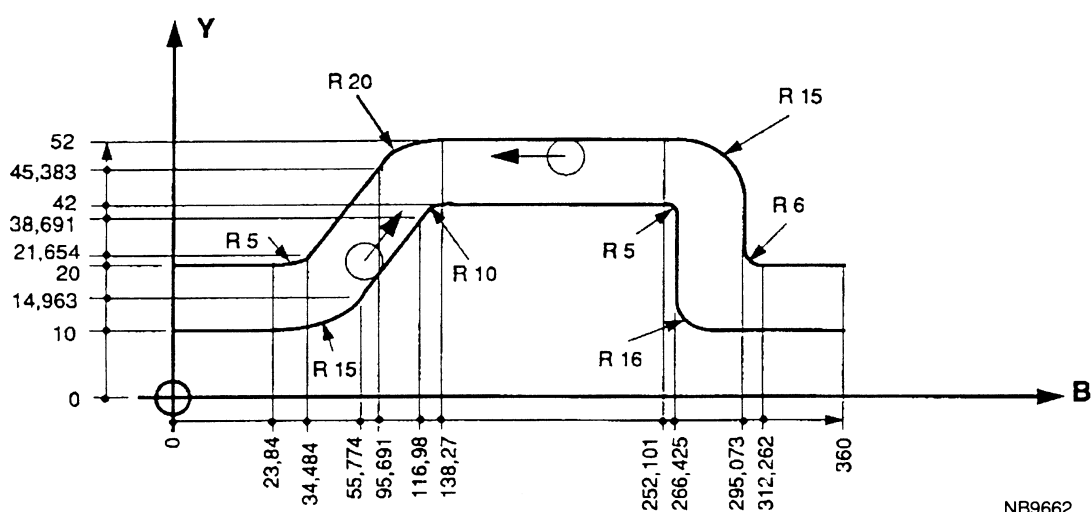
G0, G1, G2/G3, G4, G14, G22, G23, G29, G40-G44, G90/G91, G94/G95, G180/G182

- Notes:
1. If G14 or G29 are used during cylinder interpolation, the target block number for the jump must be in the program section for cylinder interpolation.
 2. The function G94 or G95, which is active at activating cylinder interpolation, is not influenced by G182.
 3. All other G-functions are not permitted, when G182 is active.

Example**EXAMPLE 1.**

The groove on the curved surface of a cylinder of diameter=40 mm has to be milled with a slotting end mill $D=9.5$ mm. The depth of operation=4 mm. The workpiece is machined horizontal with the rotary axis C, the cylinder axis Z and tool axis Y.

The contour is programmed in this example.



```
N12340
N1 G18 S1000 T1 M6
N2 G54
```

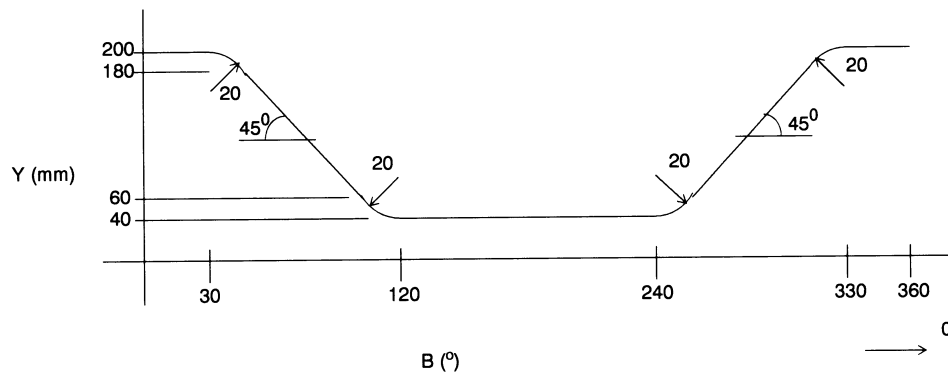
N3 G182 Y1 C1 Z1 R20
 N4 G0 Z15 C0 Y22 M3
 N5 G1 Y16 F200
 N6 G43 Z10
 N7 G41
 N8 G1 C23.84
 N9 G3 Z14.963 C55.774 R15
 N10 G1 Z38.691 C116.98
 N11 G2 Z42 C138.27 R10
 N12 G1 C252.101
 N13 G2 Z37 C266.425 R5
 N14 G1 Z26
 N15 G3 Z10 C312.262 R16
 N16 G1 C365
 N17 G40
 N18 G41 Z20
 N19 G1 C312.262
 N20 G2 Z26 C295.073 R6
 N21 G1 Z37
 N22 G3 Z52 C252.101 R15
 N23 G1 C138.27
 N24 G3 Z45.383 C95.691 R20
 N25 G1 Z21.654 C34.484
 N26 G2 Z20 C23.84 R5
 N27 G1 C0
 N28 G40
 N29 G180
 N30 G0 Y100 M30

Explanation

N1:	Load the tool. Select plane. Because the Y-axis is the tool axis, G18 must be chosen
N2:	Activate the stored zero offset of G54
N3:	Use the cylindrical coordinate system for the BY-plane, the Y-axis as tool axis and a cylinder radius of 20 mm.
N4:	Move to the start position and activate the spindle
N5:	Feed movement to depth
N6:	Radius compensation T0 the lower contour
N7:	Activate radius compensation LEFT
N8-16:	Points of the lower contour till 365° (=360° +run out)
N17:	Cancel radius compensation; the tool tip should come at the position 365°.
N18:	Activate radius compensation LEFT and move to upper contour
N19-27:	Points of the upper contour
N28:	Cancel radius compensation
N29:	Return to the basic coordinate system of the machine tool
N30:	Retract tool from part and end of program.

EXAMPLE 2.

The upper part of a groove on the curved surface of a cylinder with a radius of 114.6 mm is drawn. The workpiece is machined horizontal with the rotary axis B, the cylinder axis Y and tool axis Z. The contour is programmed in this example.



NB8558

```

N9011
N1 G17
N2 G54
N3 S500 T1 M6
N4 G182 Y1 B1 Z1 R114.6
N5 G0 Y180 B0 Z116 M3
N6 G43 Y200
N7 G1 Z114 F300
N8 G42
N9 B30
N10 G2 Y194.142 B37.071 R20
N11 G1 Y45.858 B112.929
N12 G3 Y40 B120 R20
N13 G1 B240
N14 G3 Y45.858 B247.071 R20
N15 G1 Y194.142 B322.929
N16 G2 Y200 B330 R20
N17 G1 B360
N18 G40
N19 G0 Z150 M30

```

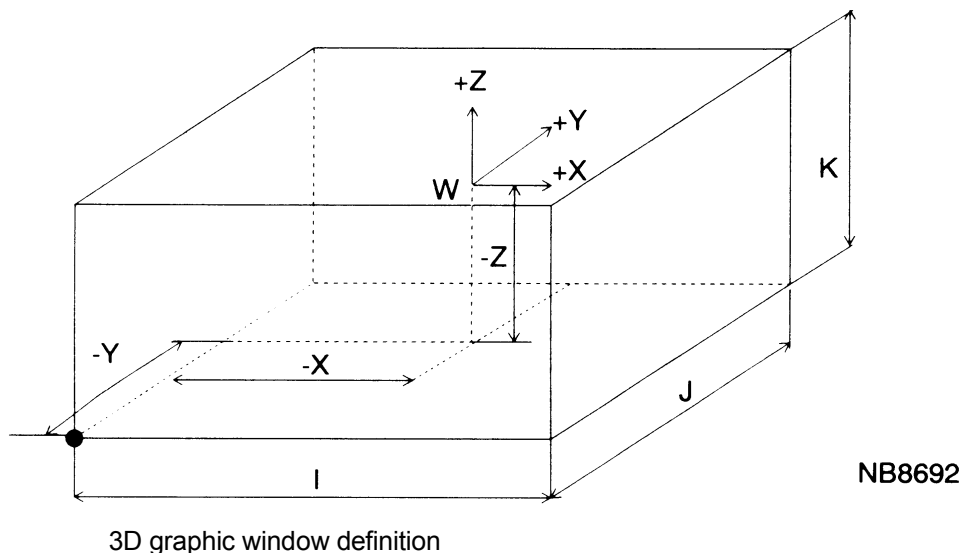
Explanation

- N1: Set the plane for operation
- N2: Activate the stored zero offset from G54
- N3: Load tool 1 and activate the spindle
- N4: Use the cylindrical coordinate system for the BY-plane, the Z-axis as tool axis and a cylinder radius of 114.6 mm.
- N5: Rapid tool movement to start position and start the spindle
- N6: Move tool to the contour
- N7: Feed movement at depth
- N8: Set radius compensation RIGHT
- N9-17: Move along the contour. The contour points must be calculated by the part programmer from the data given on the drawing.
- N18: Cancel radius compensation
- N19: Retract tool from part and end of program.

5.62 G195 Graphic window definition

Purpose

To define the dimensions of a 3D graphic window and its position relative to the zero point W. In this window the workpiece and machine parts can be situated for a graphical simulation of a part program run.



Format

N... G195 X... Y... Z... I... J... K... {B...} {B1=...} {B2=...}

Parameter

G	Graphic window definition
X	Start point coordinate
Y	Start point coordinate
Z	Start point coordinate
B	Rotation around hor. axis (3D)
I	Dimension parallel to X
J	Dimension parallel to Y
K	Dimension parallel to Z
B1=	Rotation around vert. axis (3D)
B2=	Rotation around third axis (3D)

Associated functions

G98, G99, G196 to G199

Type of function

Non-modal.

Notes and usage

GRAPHICAL SUPPORT

Refer to the appendix GRAPHICAL SUPPORT at the end of this manual for a short overview about the graphical support provided in the CNC PILOT control system and to the user manual for using the graphical support.

GRAPHIC WINDOW

The window, thus a bounded area on the display, is a rectangular 3D box which dimensions are defined by the G195-function.

The window is used with the graphical simulation, but also with the synchron graphics with which the actual tool movements on the machine can simultaneously be seen on the display of the control.

RELEVANT WINDOW AXIS

Because the display on the control is a rectangle, the scale on the shorter side (Y-axis in XY-plane), which is calculated from the programmed value (the J-word), also determines the scale on the longer axis (X-axis in XY-plane).

CONTOUR DEFINITION (G196 - G199)

Besides a window also an outer contour of a workpiece blank and/or machine parts and, if required, an inner contour can be defined for the graphical simulation. The dimensions of these contours are programmed with the functions G196 to G199. Refer to these functions for defining a contour.

DEFAULT WINDOW DIMENSIONS

If the dimensions of the 3D window are not defined the CNC uses the software limit switches' distances as default values.

ANGLE OF VIEWING (B, B1=, B2=)

With the synchron graphics or the 3D-wire plot the workpiece can be seen rotated. The angles for viewing the rotated workpiece on the display are defined by the words B, B1= or B2=.

	B rotation about	B1= rotation about	B2= rotation about
XY- plane (G17)	X- axis	Y- axis	Z- axis
XZ- plane (G18)	Z- axis	X- axis	Y- axis
YZ- plane (G19)	Y- axis	Z- axis	X- axis

Other methods are available for selecting an angle of viewing and are described in the user manual.

DEFAULT SETTINGS FOR ANGLES OF VIEWING

If the angles of viewing are not programmed the following default settings are automatically used by the control: B60, B1=30 and B2=0

RESTRICTIONS

The function G195 is not permitted in MDI or the TEACH-IN (PLAYBACK) mode.

Example

N9000	
N1 G17	Define the machining plane
N2 G195 X-30 Y-30 Z-70 I170 J150 K100	Define the graphic window
N3 G199	Start of the contour description section

5.63 G196 End contour description

Purpose

To end the contour description for the graphical simulation of a partprogram run.

Format

N... G196

Associated functions

G98, G99, G195, G197 to G199

Type of function

Non-modal.

Notes and usage

FUNCTIONS ACTIVE AFTER A G196 BLOCK

After a G196 the function G64 is reset, so G63 is activated again.

If the last function of the contour description is a G2 or G3, this function is set back to the active function G0 or G1 before the G199.

All other modal functions, which are active before the G199, are not influenced.

RESTRICTIONS

1. A G199 must be programmed before the G196. If not, an error message is displayed.
2. The G196 function cannot be used in MDI and the TEACH-IN (PLAYBACK) mode.

Example

```
N9000
N1 G17
N2 G195 X-30 Y-30 Z-70 I170 J150 K100
N3 G199 X0 Y0 Z0 D-20
...
N10 G196
```

Explanation

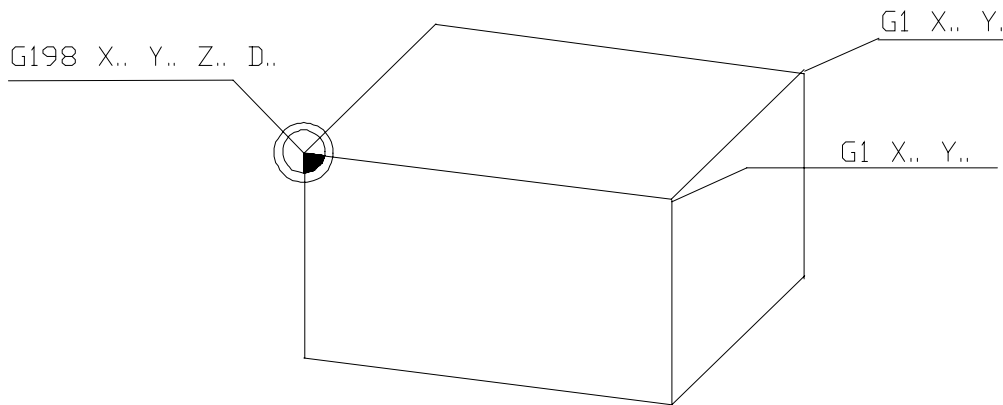
```
N1 : Define the plane of operation
N2 : Define graphic window
N3 : Start graphic contour description
:
N10 : End graphic contour description.
```

5.64 G197/G198 Begin inner/outer contour description

Purpose

To define the start point in a graphical presentation of a contour from a blank workpiece or a machine part. Outer as well as inner contours can be defined. After defining the start point of the contour the contour itself can be programmed with the functions G1 and G2 or G3.

G197: define the start point of an inner contour
G198: define the start point of an outer contour



Start point for defining the blank contour.
0: zero point defined by G199.

Format

To define the start point of an inner contour
N... G197 X... Y... D...

To define the start point of an outer contour
N... G198 X... Y... {Z...} D...

To define the start point of a geometry element:
N... G198 X... Y... {Z...} {I1=..}.

Parameter

G	Begin inside contour description
X	Startpoint coordinate
Y	Startpoint coordinate
D	Depth inside contour
I1=	Colour

G197

G	Begin outside contour description
X	Startpoint coordinate
Y	Startpoint coordinate
Z	Startpoint coordinate
D	Depth outside contour
I1=	Colour

G198

Associated functions

G98, G99, G195 to G196, G199

Type of function

Non-modal.

Notes and usage

START POINT OF CONTOUR DESCRIPTION

The start point of the contour is programmed with the linear axes coordinates X, Y and Z. These coordinates are related to the point, programmed in a G199 block.

The coordinates of the start point must be absolute, Cartesian coordinates. Polar coordinates are not allowed.

E-parameter can be used with the functions G197/G198, but parameter definition is not allowed in the G197/G198 block itself.

If the Z-word is not programmed in a G198 block, Z0 is used as a default setting.

Note: The Z-word is not used with the G197.

CONTOUR DESCRIPTION

Once the start point of the contour is established (in a G197 or G198), The contour is programmed with the functions G1, G2 and G3.

G1: a line with its end point.

G2/G3 a circular arc with end point and radius or centre point and end point.
a complete circle with centre point only.
using the helix or 2.5 D interpolation is not allowed.

An end point or centre point can be programmed with absolute Cartesian or polar coordinates. They are related to the point programmed in the G199 block.

For complicated contours the geometry of the control (G64) can be used.

The contour must be closed, otherwise, a straight line will be generated automatically from the end point to the start point.

The contour must lie in the main plane defined by the active function G17, G18 or G19.

THE DEPTH OF THE CONTOUR (D)

The depth of the outer contour is programmed with the D-word. Its value is related to the tool axis coordinate of the G198 block.

If the bottom of the contour is Z0 (XY-plane), the depth is positive. If the upper surface is Z0, the depth is negative.

The depth of the inner contour is also programmed with the D-word. Its value is related to the depth of the outer contour.

INNER CONTOURS

An inner contour must lie within a previously defined outer contour.

Inner contours may not intersect the sides or be tangent to the sides of the outer contour.

An inner contour cannot be inside another inner contour.

MORE THAN ONE CONTOUR DESCRIPTION

If a part is composed of independent contours, e.g. layers or holes, it is possible to define each contour separately with the functions G198 and G197. With one of these functions the previous contour description is ended and the description of the next contour starts.

A complete contour, thus the outer as well as the inner contour, must be programmed in one section.

If all contours are defined the function G196 is used to end the complete contour description.

DRAWING GEOMETRIC ELEMENTS (LINE OR CIRCLE)

In a G198 section a geometric element can be drawn in more colours. Spatial geometric elements will be projected on the active plane of the graphic function. So a complete drawing can be made.

This is only possible, if in a G199 block, a B4 is programmed.
The Depth (D) has no meaning and is not permitted.

Only G0, G1, G2 and G3 are permitted.

More geometric elements can be programmed in succession. In a partprogram more groups of geometric elements can be defined.

The contour is only visible during simulation in wireplot graphic.

Possible colours (I1=):

1	red	11	light red
2	green	12	light green
3	yellow	13	light yellow
4	blue	14	light blue
5	grey	15	light magenta
6	cyan	16	light cyan
7	white	17	bright white
8	black	18	black
9	foreground	19	foreground
10	background	20	background

To delete a geometric element, the same geometry in black-ground colour must be drawn.

ENDING A CONTOUR DESCRIPTION

A contour description is finished with either a function G198 for defining another outer contour, a G197 for defining an inner contour or a G196 indicating that no more contours are followed.

CONTOUR DESCRIPTION IN MACRO

If a macro is to be used for describing a contour, e.g. for describing machine parts, all graphic functions (G199, G198, G197 and G196) must be programmed in the macro.

FUNCTIONS ALLOWED WITH A CONTOUR DESCRIPTION

With a contour description only the following functions are permitted:

- G1, G2, G3: for describing a contour
- G64/G63: for defining the contour with the geometry function
- G196: for ending the contour description
- G197/G198: for defining another contour

FUNCTIONS IGNORED WITH THE CONTOUR DESCRIPTION

If radius compensation, scaling, mirror image or axis rotation is activated before the G199, the function is ignored during the graphical simulation of the contour. No error message is displayed to this effect.

Therefore it is advised to activate the following functions before the G199:

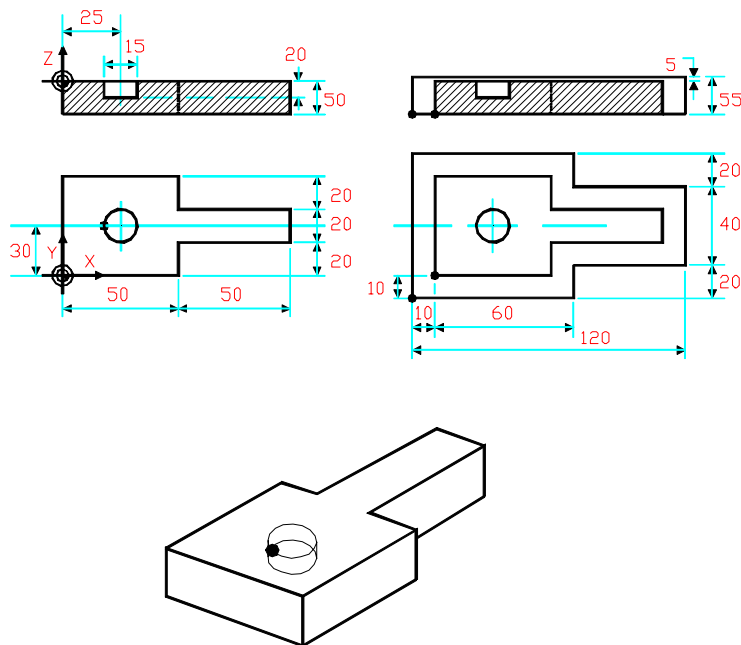
- G40 (no radius compensation)
- G72 (no mirror image or scaling)
- G90 absolute coordinates
- G93 B4=0 (no axis rotation)
- G180 Basic coordinate system

RESTRICTIONS WITH THE CONTOUR DESCRIPTION

1. The axes coordinates must be absolute and lie in the active main plane.
2. Polar coordinates and the combination of one coordinate and an angle can be used with the G1 and G2/G3 blocks. They cannot be used to define the start point of the contour.
3. Previously defined points cannot be used with a contour description.
4. The use of E-parameters to define the start point is not allowed. However, parameters can be used with the contour description itself (G1 and G2/G3 blocks).

RESTRICTIONS WITH THE USE OF G197 AND G198

1. With G197 the Z-coordinate is not used. The depth value (D-word) of the G197 is related to the depth of the outer contour.
2. The functions G197 and G198 are not permitted in MDI or in the TEACH-IN (PLAYBACK) mode.

Example**EXAMPLE 1: Outer and inner contour definition**

```

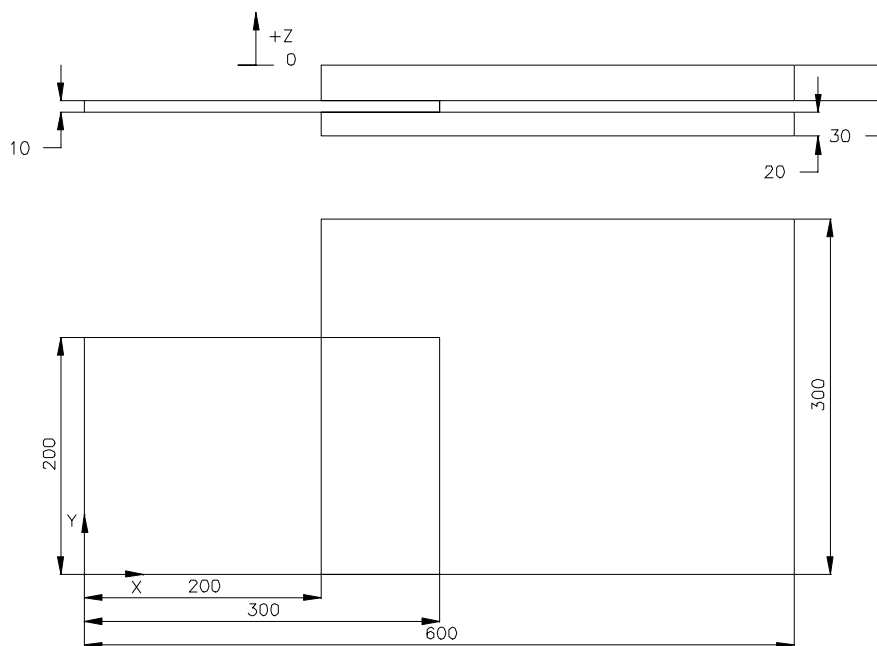
N1971981
N1 G17
N2 G195 X-10 Y-10 Z10 I120 J80 K-40
N3 G199 X0 Y0 Z0 B1 C2
N4 G198 X0 Y0 Z0 D-20
N5 G1 X50
N6 Y20
N7 X100
N8 Y40
N9 X50
N10 Y60
N11 X0
N12 Y0
N13 G197 X17.5 Y30 D-10
N14 G2 I25 J30
N15 G196

```

Explanation:

N1:	Set the plane of operation to be the XY-plane
N2:	Set the graphic window to define the 3D space
N3:	Start of contour description section.
N4:	Define start point of outer contour.
N5-N12:	Description of the outer contour. The coordinates are related to the point programmed in the G199 block.
N13:	Define start point of inner contour.
N14:	Describe the inner contour to be a complete circle.
N15:	End of contour description section

EXAMPLE 2: Contour with different layers



```

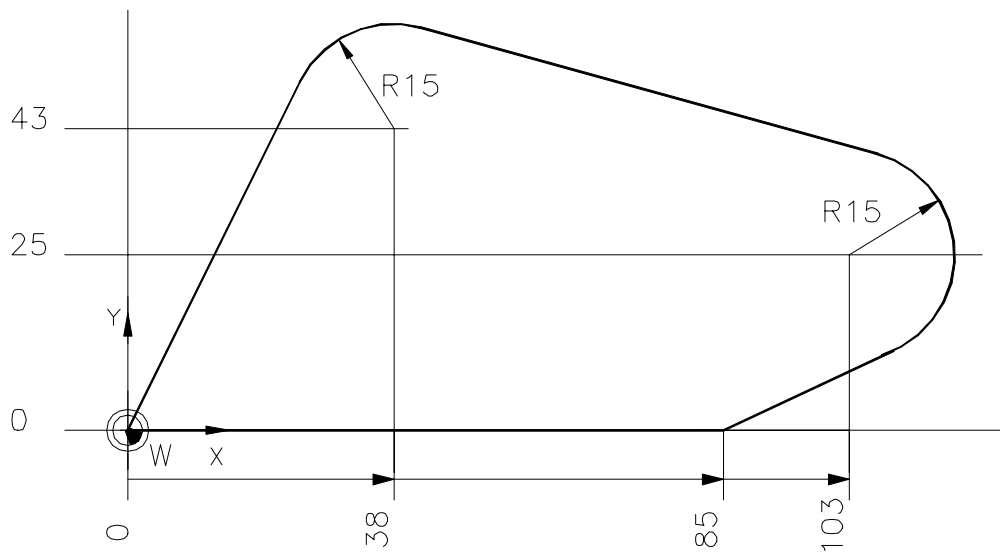
N1971982
N1 G54
N2 G17
N3 G195 X-5 Y-5 Z5 I610 J310 K-70
N4 G199 X0 Y0 Z0 B1 C2
N5 G198 X0 Y0 Z-30 D-10
N6 G1 X300
N7 Y200
N8 X0
N9 Y0
N10 G198 X200 Y0 Z0 D-30
N11 G1 X600
N12 Y300
N13 X200
N14 Y0
N15 G198 X200 Y0 Z-40 D-20
N16 G1 X600
N17 Y300
N18 X200
N19 Y0
N20 G196

```

Explanation:

N1:	Set the program zero point
N2:	Set the plane of operation to be the XY-plane
N3:	Set the graphic window to define the 3D space
N4:	Start of contour description section.
N5:	Define start point of outer contour (first layer).
N6-N9:	Describe the first layer
N10:	Define start point of outer contour (second layer).
N11-N14:	Describe the second layer
N15:	Define start point of outer contour (third layer).
N16-N19:	Describe the third layer
N20:	End of contour description section

EXAMPLE 3: Contour described with geometry



```

N1971983
N1 G54
N2 G17
N3 G195 X-10 Y-10 Z10 I138 J78 K-30
N4 G199 X0 Y0 Z0 B1 C2
N5 G198 X0 Y0 Z0 D-10
N6 G64
N7 G1 X85 Y0
N8 R1=0
N9 G3 I103 J25 R15 R1=0
N10 G1 R1=0
N11 G3 I38 J43 R15 R1=0
N12 G1 X0 Y0
N13 G63
N14 G196
N15 S1000 T1 M6
N16 G0 X-15 Y-15 Z20 M3
N17 G1 Z-10 F1000
N18 G43 Y0
N19 G42
N20 G14 N1=6 N2=13
N21 G40
N22 G0 X-15 Y-15 Z100 M30

```

Explanation:

N1:	Set the program zero point
N2:	Set the plane of operation to be the XY-plane
N3:	Set the graphic window to define the 3D space
N4:	Start of contour description section.
N5:	Define start point of outer contour.
N6-N13:	Describe the outer contour. The coordinates are related to the point programmed in the G199-block. The geometry of the control is used to program the contour.
N14:	End of contour description section

N15:	Load the tool and set the spindle speed
N16:	Start the spindle and move the tool to the start point.
N17:	Move tool to depth
N18:	Move tool to the contour
N19:	Set radius compensation RIGHT
N20:	Move along the contour. With the G14 function the programmed contour in the outer contour description is used.
N21:	Cancel radius compensation
N22:	Move tool to the start point and away in the tool axis. End of program

EXAMPLE 4: Draw a raw material (cylinder) for turning.

```

N9999
N1 G17
N4 G36 (endless turning)
N6 G17 Z1=1 Y1=2
N7 G195 X-1 Y-1 Z1 I2 J12 K-11
N8 G199 X0 Y0 Z0 B4 C2
N9 G198 I1=14 X0 Y8 Z0
N10 G2 X0 Y0 I0 J0
N11 G1 X0 Y8 Z-8
N12 G2 X0 Y8 I0 J0
N13 G196
N14 ...

```

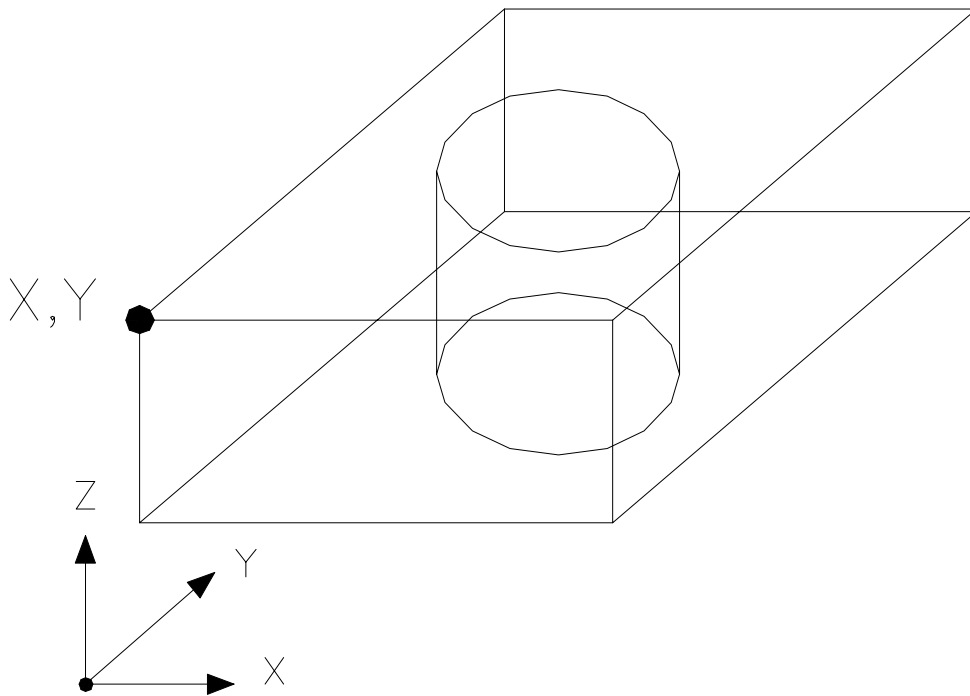
Explanation:

N1	Set the plane for milling. Length compensation in Z-direction.
N4	Turning mode
N6	Set the plane for turning. Mainaxis-1 is Z, Mainaxis-2 is Y. Radius compensation in ZY plane.
N7	Set the graphic window
N8	Start of contour description section. B4 means draw.
N9	Define start point of contour. I1=14 means colour pale blue
N10	Upper circle of the cylinder.
N11	Connection line
N12	Lowest circle of the cylinder.
N13	End of contour description section

5.65 G199 Begin contour description

Purpose

1. To define the position of the workpiece blank contour related to the program zero point or machine zero point. This position is used during the graphic simulation of the program run.
2. To define the position of any machine part with which the tool might collide. Collision to be detected during the graphic simulation.
3. Drawing a contour during the wireplot simulation.



4. To draw one or more geometry elements (line or circle) during the wireplot simulation.

Format

To define a blank contour

N... G199 [Coordinates of position] B1 {C1} {C2}

To define a machine part

N... G199 [Coordinates of position] B2 {C1} {C2}

To draw a contour during the wireplot simulation.

N... G199 [Coordinates of position] B3 {C1} {C2}

Draw one or more geometry elements (line or circle) during the wire model graphic simulation.

N... G199 [co-ordinates of position] B4 {C1} {C2}

Parameter

G	Begin graphic model description
X	Startpoint coordinate
Y	Startpoint coordinate
Z	Startpoint coordinate
B	1=mat., 2=mach., 3=contour, 4=draw
C	Zero point 1=Machine, 2=Workpiece

Associated functions

G98, G99, G195 to G198

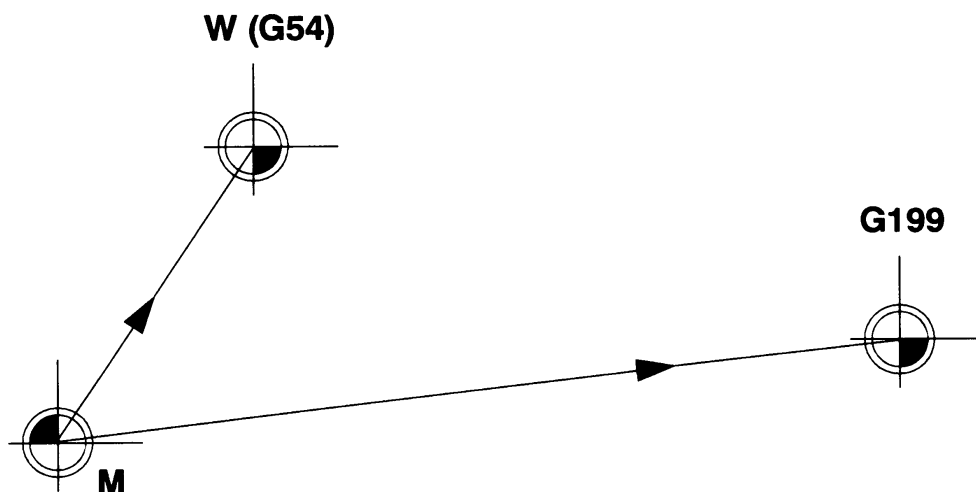
Type of function

Non-modal.

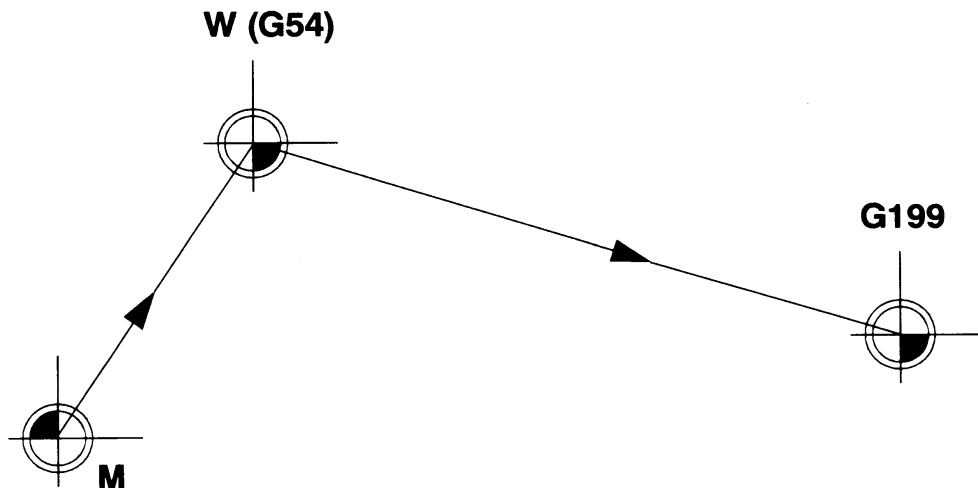
Notes and usage

ZERO POINT (C1/C2)

The position of the contour can be related to either the machine zero point M_0 or the program zero point W .



C1 the contour position is related to the machine zero point M_0 .



C2 the contour position is related to the program zero point W.

DEFAULT ZERO POINT

If the C-word is not programmed the following default settings are used:

For a workpiece blank the G199 coordinates are related to the program zero point W (C2).

For machine parts the G199 coordinates are related to the machine zero point M_0 (C1).

COORDINATES OF CONTOUR POSITION

The coordinates of the contour position must be absolute, Cartesian coordinates. Polar coordinates are not allowed. These coordinates are related to the point defined with the C-word

Defining the position with E-parameters is not possible. Also defined points cannot be used. The position of the contour is related to the program zero point W.

TYPE OF CONTOUR DESCRIPTION (B)

A contour description can be used for four purposes:

- 1 to define a blank contour (uncut part) (B1=)
- 2 to define machine parts, such as clamping devices, etc. (B2=)
- 3 to draw a contour of a machined work piece. (B3=)
4. to draw geometry elements (Line or circle) (B4=).

WORKPIECE BLANK (B1)

The outer and inner contour of a workpiece blank, thus the form of the uncut material, can be described with the graphic functions. The cutting of this material to the required shape can be simulated on the control.

Refer to the functions G197/G198 for details about describing a contour.

MACHINE PARTS (B2)

The contours of machine parts, such as clamping devices, can also be described by using the G199 function. This allows possible collisions between the cutting tool and machine parts to be detected.

The details given for the description of workpiece contours also apply to machine parts, except the B-word in the G199 block, which must be B2.

The description of machine parts is usually at the beginning of the partprogram. However, the description blocks can also be placed later in the program, in a particular part of the machining cycle.

A number of shapes can be programmed in succession enabling a machine part to be made up of several layers.

Several machine parts can be defined in a part program.

With macros a library for machine parts can be built up.

CONTOURS (B3)

The contours of a machining part (outside contour rectangular pocket) can be describes with the graphical function.

The contour is only visible in wire plot simulation.

For special information about the contour description see the functions G197/G198.

The special information of B1 and B2 is also valid for the B3.

The contours can be describes all over in the program. More contours are possible.

There are no differences between G198 and G197 contours.

The contours are be drafted in the colour Cyan.

GEOMETRIC ELEMENTS (LINIE OR CIRCLE) (B4)

Geometric elements (line or circle) can be drawn with the G198-functionality. Spatial geometric elements will be projected on the active plane of the graphic function. So a complete drawing can be made.

With G199 B4 must defined that geometric elements will be drawn

The contour is only visible during simulation in wireplot graphic.

For description of the geometric elements refer to function G198.

TOOL IMAGE

A tool image can be assigned to the tool with the aid of the G-word in the tool memory. The required image can be selected from a set of available tool images and is used by the CNC system to accurately simulate the machining.

Refer to the user manual for selecting the tool image.

GRAPHIC FUNCTIONS AND MACROS

The function for starting the contour description (G199), the contour description itself (G198 and / or G197) and the function for ending it (G196) must be in the same partprogram or subprogram (macro).

E-parameter can be used with the function G199, but parameter definition is not allowed in the G199 block itself.

ANOTHER BLANK CONTOUR DESCRIPTION

In a part program more than one contour for a blank workpiece can be described. Each contour must be defined with the functions G199, G198, G197 and G196. A new graphic window (G195) must be defined too.

Only one blank contour is shown on the display. As soon as another description is encountered, the previous one is deleted and the other model shown. The tool movements are shown in the displayed model.

PLANE SELECTION (G17, G18, G19)

If a function for plane selection (G17, G18, G19) is encountered, the displayed program is deleted. After the G-function for plane selection another contour description can be given. The programmed movements are displayed in this model.

Only the movements in the last programmed plane are shown.

CYLINDER INTERPOLATION

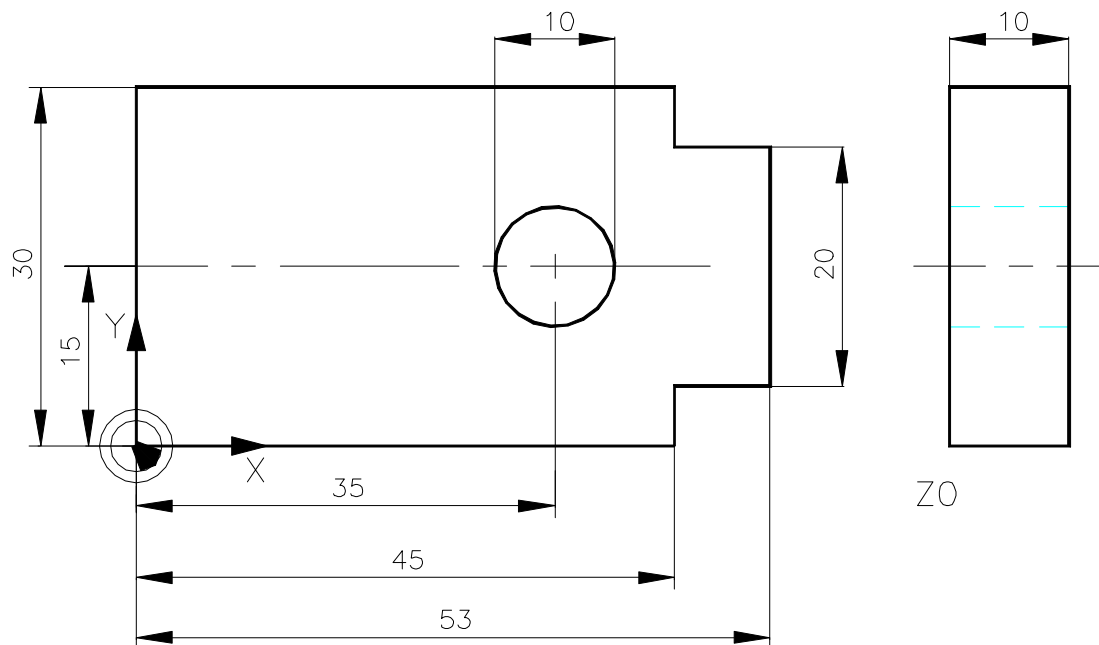
Movements programmed in the plane of the cylinder (G182) cannot be visualized on the display of the control.

RESTRICTIONS

1. A G198 function must immediately follow the G199 block.
2. The G199 function cannot be used in MDI or TEACH-IN (PLAYBACK) mode.
3. A contour description should not be repeated with a G14 at another place. In this case the control repeats the model and the operations executed in this model before the G14 and does not stop properly. It is advised to write the blocks again in the program or to put the contour description in a macro.
4. A contour description is only allowed in the basic coordinate system, thus G180 active.

OPERATION WITH A CONTOUR DESCRIPTION

1. A described contour between a G199 and a G196 is treated as one block. This means that at SINGLE BLOCK the complete contour is executed in one step.
2. A MANUAL BLOCK SEARCH to a block between a G199 and a G196 is not possible and results in an error message.
3. INTERVENTION during the drawing of the contour is not possible. The INTERVENTION is executed once the complete contour is drawn.

Example**EXAMPLE 1.**Defining a blank contour with clamping devices

Each clamp is described in a separate macro. With two parameters the start point of the clamp contour is programmed:

- E1: X-coordinate of the contour start point related to the program zero point.
- E2: Y-coordinate of the contour start point related to the program zero point.

The contour itself is programmed with fixed dimensions.

If this clamp is used in different part programs, the two parameters must be set at the macro call and then the clamp can be used for graphical purposes.

Macro for the left clamp

```

N1991
N1 G92 X=E1 Y=E2
N2 G199 X0 Y0 Z0 B2 C2
N3 G198 X0 Y0 Z0 D10
N4 G1 X45
N5 Y5
N6 X53
N7 Y25
N8 X45
N9 Y30
N10 X0
N11 Y0
N12 G197 X30 Y15 D-10
N13 G2 I35 J15
N14 G196
N15 G92 X=-E1 Y=-E2

```

Macro for the right clamp

```

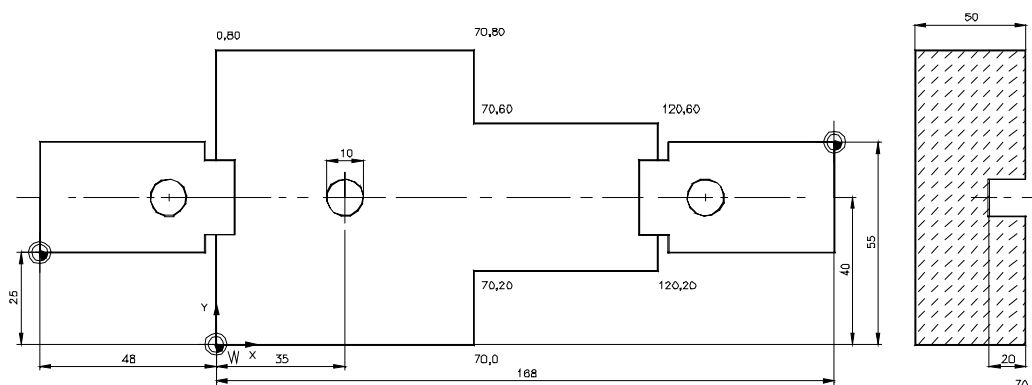
N1992
N1 G92 X=E1 Y=E2
N2 G199 X0 Y0 Z0 B2 C2
N3 G198 X0 Y0 Z0 D10
N4 G1 X-45
N5 Y-5
N6 X-48
N7 Y-25
N8 X-45
N9 Y-30
N10 X0
N11 Y0
N12 G197 X-30 Y-15 D-10
N13 G2 I-35 J-15
N14 G196
N15 G92 X=-E1 Y=-E2

```

Explanation of the macros

- N1: A zero point shift to let the start point of the clamp coincide with the program zero point.
- N2: Start the contour description section of the clamp. Its zero point is indicated in the drawing. Coordinates related to the program zero point are used.
- N3: Start of the outer contour description of the clamp. Offset values relative to point stated in G199-block.
- N4/N11: Describe the outer contour. Coordinates are related to the point defined in the G199-block. The depth D is measured from the surface (Z0 in G199).
- N12: Start of the inner contour description. Offset values relative to point stated in G199-block. The depth D is measured from the surface (Z0 in G199).
- N13: Describe the inner contour to be a hole. Centre point coordinates are related to the point defined in G199-block.
- N14: End of the description section of the clamp.
- N15: Restore the program zero point.

EXAMPLE 2. The graphic section of the part program:



```

N199000
N1 G17
N2 G54
N3 S1200 T1 M6
N4 G195 X-20 Y-20 Z-60 I160 J110 K70
N5 G199 X0 Y0 Z0 B1 C2
N6 G198 X0 Y0 D-50
N7 G1 X70
N8 Y20
N9 X120
N10 Y60
N11 X70
N12 Y80
N13 X0
N14 Y0
N15 G197 X30 Y40 D-20
N16 G2 I35 J40
N17 G196
N18 G22 N=1991 E1=-48 E2=25
N19 G22 N=1992 E1=168 E2=55
:
N200 M30

```

Explanation:

N1:	Set the main plane to be the XY-plane
N2:	Set the zero point
N3:	Load the tool and set the spindle speed
N4:	Set the graphic window to define the 3D space
N5:	Start the contour description section of the workpiece blank. The start point of the contour coincides with the program zero point.
N6:	Start of the outer contour description. Offset values relative to point stated in G199-block.
N7/N14:	Describe the outer contour. Coordinates are related to the point defined in the G199-block. The depth D is measured from the surface (Z0 in G199).
N15:	Start of the inner contour description. Offset values relative to point stated in G199-block. The depth D is measured from the surface (Z0 in G199).
N16:	Describe the inner contour to be a hole. Centre point coordinates are related to the point defined in G199-block.
N17:	End of the description section of the blank contour.
N18:	Call the macro for the left clamp
N19:	Call the macro for the right clamp

5.66 G200---G208 Pocket Cycle

5.66.1 G200 Begin pocket cycle

Purpose

The Universal Pocket Cycle allows easier and quicker production of CNC programs, which control the milling of rectangular and circular pockets. 'Islands' of uncut material within the pockets can also be produced.

Format

N... G200

Associated functions

G200...G208

Parameter

Non

5.66.1.1 Introduction universal pocket cycle

The Universal Pocket Cycle (UPC) allows easier and quicker production of CNC programs, which control the milling of rectangular and circular pockets. 'Islands' of uncut material within the pockets can also be produced.

The CNC can calculate the minimum number of starting points to produce the required pocket contours in the shortest time. Tool movements are based on using the contour-parallel method, which is the most effective technique for machining pockets.

A 'start point' macro can be generated and used to pre-drill the cutter start position(s) in the workpiece. The minimum number of start points required to machine the pocket are calculated, keeping machining times to a minimum.

The programmer only needs to state the program numbers of the required UPC macros (subprograms), together with the machining parameters such as the feed rate, clearance distance, cutter radius etc., and the start point and dimensions of the pocket contour.

The CNC uses the above data to calculate the start points and coordinates of the toolpaths, which will be parallel to the contour's sides. These calculations are performed before the pockets are milled. A UPC macro is calculated only once; the contents of this type of macro will be re-used whenever the macro is reactivated in the CNC program.

The macro is calculated only once. The macro contents are reused every time the macro is called in the CNC program. If changes have been made (e.g. different cutter), the macro is recalculated.

The tool movements proceed to a contour-parallel process, which enables milling pockets to be machined most economically.

A 'finishing' cutting path can be incorporated in a machining cycle as a separate UPC macro.

This function must be programmed before the universal pocket cycles that have to be calculated, and commands that:

- if the calculations have not yet been made, the coordinates of the cutter paths must be calculated
- the cutter paths will be included in a macro generated by the CNC; the number (N1=...) of this machining macro is programmed in the G201-block
- if necessary (stated in a G201-block, by N2=...), a second macro will be generated for drilling the starting points
- if necessary (stated by the G203- or G205-function), the macros (N1=...) for finishing the contours will be generated.

All operational conditions such as the machining plane, zero point shifts and tool offsets, must be active before the G200-function is executed.

Point definitions (G78), which will be used for the pocket contour definition, must be defined before the G200-block.

All universal pocket cycles, which have been programmed between a G200- block and G202 or M30, will be calculated (if necessary).

A G200-block may be included in a macro, however the pocket will only be searched for in macros, which are nested deeper.

The CNC calculates the UPC macros before executing the CNC partprogram. Therefore, any blocks between G200 and G201 will initially be ignored. After the macros have been generated, these blocks will then be executed.

The generated macros are in the macro memory not visible for the operator. If the macro must be used in another program, the macro number must be entered in the macro memory. Then the macro is visible in the macro memory and the macro can read in and out.

For reasons of memory space the pocket contour and associated islands may not exceed 50 sides. 15KB of additional memory capacity will be necessary to allow approximately 50 sides to be stored. A circular movement that is greater than, or equal to, 180E will be treated as if it had two sides because the CNC system automatically divides this type of movement into two equal parts.

The machining plane (G17/G18/G19) must be selected before the G200-function or after a G202-function, is executed.

If the coordinates of the defined points are changed after the pocket has been calculated, and if the pocket must be calculated again, the macros generated during the calculation of the second pocket will be stored. The macros of the first calculation will be destroyed.

For calculating the macros, characteristics and quantities such as programmed points, scaling, rotations, mirror image etc. will be used the way they are active with the G200-block concerned.

If errors are generated in a pocket cycle program or graphics shows material left, the programmer is advised to make alterations to the program in order to prevent those errors. Proposals are changing the overlap percentage, change-programming sequence, change entry point, split up program, and define pseudo-island of 1 micron.

If a calculation error occurs during execution of a pocket cycle (0170, 0176) please check the pocket contour. If the contour seems ok check the tool radius size with respect to contour and islands, and choose a tool with a smaller radius when the originally programmed tool is comparatively large.

If in pocket cycles the tool radius is relatively large compared to the pocket or its islands, some problems may occur:

- G201 I Certain I-values may lead to material left or damage of islands programmed.
- Errors P163, 0170 or 0176 could be set.

The only cure in above cases is to choose a smaller tool for machining this pocket cycle concerned.

5.66.1.2 Part program structure

The example below shows a simple program, which uses a macro, which specifies a pocket cycle.

```

N99999 G54
N1 G17
N2      \
:      > Normal machining
N96    /
N97 G200           Begin pocket cycle
N98 G81           Drill cycle description
N99 G22 N=..      Start point drilling
N100 G201 N1=.. N2=.. Starting to mill pocket cycle
N101 G203 N1=..      Begin pocket contour description
N102  \
:      > Pocket contour description
N109  /
or
N101 G208 N1=..      Pocket contour description of a regular quadrangle.
N110 G204           End pocket contour description
N111 G205 N1=..      Start island contour description
N112  \
:      > > Island 1 contour description
N118  /
N119 G206           End island contour description
N120 G205 N1=..      Start island contour description
N121  \
:      > Island 2 contour description
N129  /
N130 G206           End island contour description
N131 G207 N=..       Island contour description 3 is a macro
N132 G202           End of pocket cycle

N350 G22 N=..      Pocket contour finishing
N351 G22 N=..      Island 1 finishing
N352 G22 N=..      Island 2 finishing
N353 G22 N=..      Island 3 finishing

```

5.66.1.3 Translation, rotation and mirror image of a pocket

A pocket cycle can be described using a datum point which is different from the datum point used during program execution; this may be necessary for either arithmetical reasons, or if the pocket is included in a macro.

A different datum point is established by programming a datum point shift and/or axis rotation before the G201-block.

The datum point shift and/or axis rotation are programmed with the standard function G92 or G93. The axis transformation will then be performed on the generated machining macro.

When the pocket has been cut the programmer must ensure that the program datum point is reset at the correct location. Similarly, when using the macro for the starting points and macros for finishing the pocket contour, the programmer must ensure that the correct datum point is used.

A pocket may be used again within the same part program by programming a new start position and orientation.

Example:

```

N9001
:
N90 G200
N100 G201 N1=9999
N110 G203
: \
: > Description of the pocket contour
: /
N200 G204

N205 G205
: \
: > Description of an island contour
: /
N206 G206
N300 G202
:
N400 G92 X.. Y.. Z.. B4=..
N410 G22 N=9999
:
N500 G73 X-1
N510 G22 N=9999

```

Explanation:

In block N400 the datum point is shifted and the pocket rotated (B4=..), so that the starting point of the pocket is correctly positioned. In block N410 the machining macro is called. The pocket will be cut again but at a different location.

In this example (with G17 active), mirroring about the Y-axis is programmed in block N300 and the machining macro executed in block N310 as mirror image. The disadvantage is, that one pocket is machined in a backward direction and the other in a forward direction. When this is not possible for technological reasons, the mirror image function cannot be used. The mirrored pocket must then be programmed again.

5.66.1.4 Same pocket in another program

If a pocket (and associated islands) occurs in different programs, the complete pocket may be written in a macro (subprogram); this macro will then be called at that point in the partprogram where the pocket must be cut.

The functions G201, G202, G203/G204, and G205/G206 have to be stated in the macro.

Programming will be:

```

N9001 G201 Y.. Z.. B.. N1= N2=
N1 G203
: \
:   > Pocket contour
: /
N8 G204
N9 G205
: \
:   > Contour of island 1
: /
N13 G206
N14 G205
: \
:   > Contour of island 2
: /
N18 G206
N19 G202

```

Represents a macro of a pocket, which has islands.

A partprogram, which uses this macro, could look like:

```

N9999
N1 G200
:
N50 G22 N=9001
:

```

A datum point shift prior to the macro call positions the pocket in the correct location.

Remark: By including the G201-block in a macro, the nesting level of the machining macro will be increased by one. Macros cannot be nested more than 8 times.

5.66.1.5 Operating section**MACROS FOR POCKETS, STARTING POINT AND FINISHING MACROS****GENERATION**

When the control encounters a G200-block in a program, it searches for the pocket functions (G201-block + associated contour description + G202-block). Upon finding a G201-block (in the partprogram or possibly via a macro call) and if no macro carrying the programmed macro number (N1=.. in the G201-block) exists, a pocket macro will be generated by the control. (The data, which determine the pocket macro, viz. the parameters Y, Z, B, R, I, K in the G201-block and the contour description, will be stored with the pocket macro).

If a macro with the programmed pocket macro number exists already, the associated data will be compared with the programmed data. If they do not match, a new pocket macro will be generated.

A new starting points macro is generated if:

a starting points macro number is programmed (N2=.. in the G201-block) and a new pocket macro is generated or if no macro with the programmed starting points macro number exists.

Finishing macros are generated if:

a finishing macro number is programmed (N1=.. in the G203- or G205-block concerned) and a new pocket macro is generated or if no macro with the programmed finishing macro number exists.

REMOVAL

After being generated the macros are "locked". In order for them to be removed they must first be "unlocked". Removal may be necessary, for example, if a G201-block is being removed from a program or if an 'N1=' or 'N2=' address in a G201-block or an 'N1=' address in a G210-block is being changed.

The macros will be automatically removed if a program is started in which the pocket definition has been changed.

In V330 are for the operator, the generated macros in the macro memory no longer visible. For using a macro in another program, the macro number must be given in the macro memory. After that the macro will be visible in the macro memory. It is possible to read-in and read-out the macro program.

SIGN OF LIFE

The Universal pocket cycle software is designed to stop all calculations upon detection of an error. To indicate that the CNC is busy a rotating indicator, with the word "CLOCK", is displayed on the screen. While the indicator is rotating the CNC is busy calculating the pocket cycle.

TEACH-IN

The use of a pocket G-function (G200 to G208) in TEACH-IN MDI and TEACH-IN/PLAYBACK is not permitted.

EDITING

During editing, static or dynamic programming support may be used.

A (new) program may be edited while a program with a pocket section is running.

BLOCK DELETE

If a G201-block contains a "/" character and "block delete" is activated, then:

G200 will be executed in the normal way, i.e. macros will be generated if necessary. The G201-block, the contour description and the G202-block will be skipped during the program execution, i.e. program execution will continue after the G202-block.

BLOCK SEARCH

Searching a G200-block causes pocket macros to be generated if they are not yet present.

During program execution (after the G200-block has been executed), G201 is taken as a macro call (G22) to the pocket macro. However, after executing the pocket, a jump is done to the first block after the terminating G202-block.

Searching for a block in a pocket macro is performed in the same way as searching for a block in a macro called via G22.

Note: As G201 is taken as a macro call during program execution, the nesting level (max. 8 levels) should be taken into account when programming G201.

INTERVENTION

Intervention during the execution of a G200-block is possible. However, macros have to be recalculated after an intervention and this could take a long time if a complex contour is required.

Execution can be resumed via "start". After intervention, it is not possible to enter the edit mode without a "clear control".

Intervention during the execution of a pocket macro is dealt with in the same way as intervention in a macro called with G22.

INCOMPLETE PROGRAMMING

Normally, the following G-functions are programmed for a pocket cycle: G200, G201, G202, G203/G204, G205/G206, G207 and if necessary, G208.

G201 and G202, G203/G204 and G205/G206, are required to appear as a combination in the same program or the same macro, otherwise an error message is issued.

If a G203/G204 or G205/G206 combination appears (possibly via a macro call G22) without the associated G201/G202 combination, an error message is issued.

A G207 should always appear after a G205/G206 combination, with both G205/G206 and G207 belonging to the same G201-block, otherwise an error message is issued.

If a G201/G202 combination is programmed without contour descriptions, an error message is given.

If G200 is programmed without G201/G202 contour descriptions, the program is executed in the normal way (without pocket cycle).

If G201/G202 contour descriptions have been programmed without G200, no macros are generated and G201 is taken as G22,

OPERATION MODE CHANGE

An operational mode change can only occur after a G200-block has been completely executed.

5.66.1.6 Error messages

Number	Message
--------	---------

#5	Memory pool exhausted
O170*	Pocket-cycle calculation error (Operating error).
Solution:	Decrease number of parameters (MC 83).

O176*	Module xxx, number xxx.
-------	-------------------------

Note: The 0170 and 0176 messages are displayed together. Solving this problem can be achieved by using one of three methods:

- Change the tool radius.
- Change the overlap I-word value
- Change the amount of stock to be left for the finishing cycle

P07 Programmed data is out of range

In a block with a circular (G2/G3) side programmed with a radius value. When the programmed radius is two times greater than the distance between the start and end point. This can occur on a side of 180°.

The error is removed by changing the start or end point.

P35 End point not on circle.

On a block with a circular (G2/G3) side programmed with centre point or a block with a circular (G2/G3) side programmed with geometry. The value of the programmed centre point is greater than the value of the Circular end point window (calculated centre point between start and end point) specified by machine constant 712.

The error can be removed by changing the centre point or the start/end point.

P75 Circle without centre point

An arc has been programmed which has the same start and end point.

The error is removed by deleting the program block or by programming an arc, which has a separate start and end point.

P140 Invalid G207 nesting

P141 Too many sides programmed

P142 Too many contours programmed

P143 Invalid 'G' in pocket-cycle mode

P144 Invalid contour description

This error is caused by one of the following reasons:

- - The G208-function has been omitted
- - The G208-function specifies rounding (radii) which are too large
- - The G208-function block contains X=0 or Y=0
- - The G208-function block contains B1 > 180°
- - The G208-function block contains an I or R-word

P145 Invalid start point for approach.

A finishing cutting path cannot be generated from the current description, another start point must be chosen.

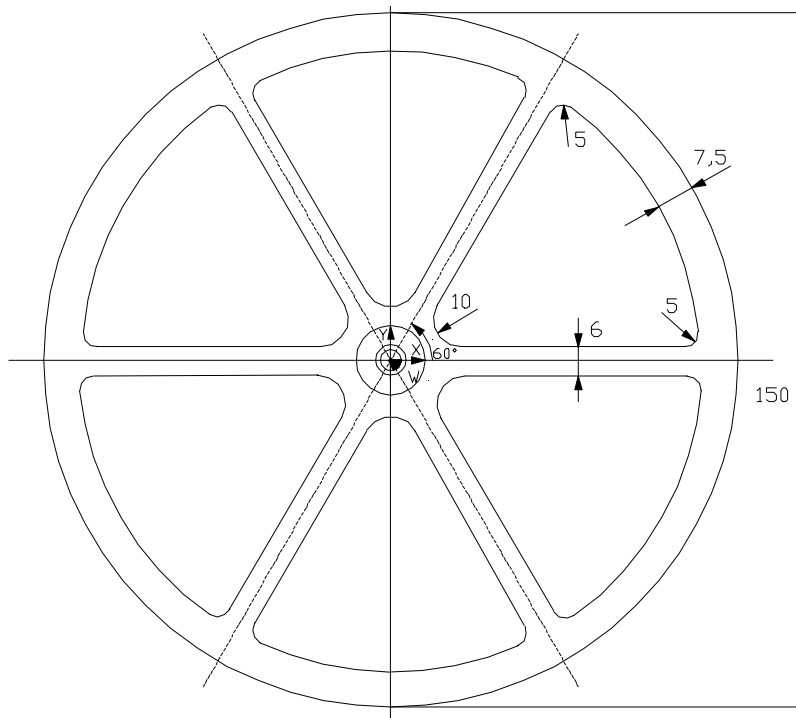
P146 No G202 defined
P147 Memory manager error
P148 Floating point error
P150 Tool not found
P160 Pocket macro generation error
P161 Finishing macro generation error
P162 Macro start point generation error

Note: Errors P160, P161 and P162 are produced by:
- a full CNC memory
- Too many start points having to be calculated

P170 Contour xx not closed
P171 Contour xx has more inner areas
P172 Contour xx intersects contour xx
P173 Contour xx enclosed by contour xx
P174 Contour xx is outside the pocket

Example

Rotated pocket



N3620511 (WHEEL AS POCKET)

N1 G17

N2 G54

N3 G195 X-90 Y-90 Z0 I180 J180 K-10

N4 G99 X-85 Y-85 Z0 I170 J170 K-10

N5 G200

N6 T31 M6 (DRILL RADIUS 4. mm)

N7 G81 Y1 Z-5 F100 S100 M3

N8 G22 N=3620501

N9 G92 B4=60

N10 G14 J5 N1=8 N2=9

N11 G93 B4=0

N12 T04 M6 (ROUGHING MILL RADIUS 3. mm)

N13 S1500 M3

N14 G201 Y0.1 Z-5 B1 I50 F1000 N1=3620500 N2=3620501 F2=500

N15 G203 X37.5 Y3 Z0 N1=3620502

N16 G64

N17 G1 X1=0 Y1=3 B1=0 J1=2

N18 G3 R5

N19 I0 J0 R67.5 J1=1

N20 R5

N21 G1 X1=0 Y1=0 B1=-120 I1=-3

N22 G3 R10

N23 G1 X37.5 Y3 B1=0

N24 G63

N25 G204

N26 G202

N27 G92 B4=60

N28 G14 J5 N1=14 N2=27

N29 G93 B4=0


```
N30 T3 M6 (FINISHING MILL RADIUS 2.5 mm)
N31 S1800 M3
N32 G22 N=3620502
N33 G92 B4=60
N34 G14 J5 N1=32 N2=33
N35 G0 Z100 M30
```

Explanation:

The CNC processes the blocks N3 and N4, which define a graphic simulation of the program's operation.

The UPC macros are calculated first (blocks N5 and N14 to N26 are executed). The following macros are created:

- Macro No. 3620500 for the machining-cycle;
- Macro No. 3620501 for the starting-points;
- Macro No. 3620502 for the finishing-cycle.

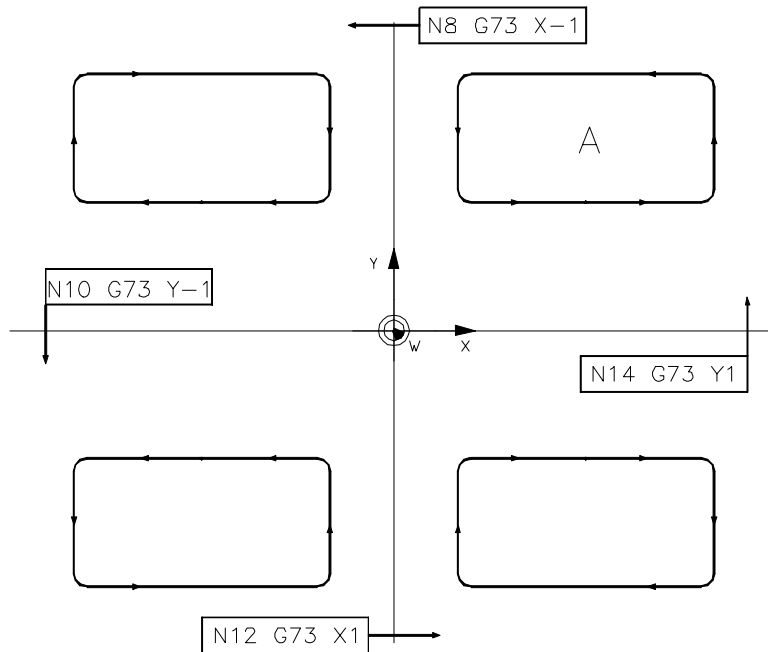
Tool 31 is selected and a drilling cycle defined by the function G81. CNC control is transferred to UPC macro no. 3620501 by the use of E-parameter no.1 and the G22-function. The first starting points are drilled. Block N9 commands the macro's coordinates to be rotated 60° in a counter clockwise direction.

Tool 4 is selected. Block N14 commands that the machining cycle (macro no. 3620500) is executed. When the machining cycle is finished, the coordinates of the macro 3620500 are rotated 60° in a counter-clockwise direction. Block N28 commands the machining cycle to be repeated and rotated five times (Note: N2=27 can be omitted from the block and the repeat instruction will still operate).

Tool 3 is selected. Block N32 assigns the value 3620502 to E-parameter no.1. The 'finishing' cycle is executed in the same manner as for the two previous UPC macros.

Block N35 commands the tool to move to Z100 and the partprogram to end.

Mirrored pocket



A: Programmed pocket cycle

N3620513 (MIRROR IMAGE OF A POCKET)

N1 G17

N2 G54

N3 G195 X-150 Y-110 Z0 I300 J220 K-10

N4 G99 X-145 Y-105 Z0 I290 J210 K-10

N5 G200

N6 T31 M67 (DRILL RADIUS 4. mm)

N7 G81 Y1 Z-5 F500 S1000 M3

N8 E1=3620501

N9 G22 N=E1

N10 G73 X-1

N11 G14 N1=9

N12 G73 Y-1

N13 G14 N1=9

N14 G73 X1

N15 G14 N1=9

N16 G73 Y1

N17 T4 M6 (ROUGHING MILL RADIUS 3. mm)

N18 S1800 M3

N19 G201 Y0.1 Z-5 B1 I60 N1=3620500 N2=3620501 F1000 F2=500

N30 G203 X75 Y50 Z0 N1=3620502

N21 G1 X120

N22 G3 X125 Y55 R5

N23 G1 Y95

N24 G3 X120 Y100 R5

N25 G1 X30

N26 G3 X25 Y95 R5

N27 G1 Y55

```

N28 G3 X30 Y50 R5
N29 G1 X75 Y50
N30 G204
N31 G202
N32 E1=3620500
N33 G14 N1=10 N2=16
N34 T3 M6 (FINISHING MILL RADIUS 2.5 mm)
N35 S2000 M3
N36 E1=3620502
N37 G14 N1=9 N2=16
N38 G0 Z100 M30

```

Explanation:

The CNC first processes blocks N3 and N4, which defined a graphics simulation of the partprograms operation.

The UPC macros are calculated first (blocks N5 and N19... N31 are executed). The following macros are created:

- Macro No. 3620500 for the machining-cycle;
- Macro No. 3620501 for the starting-points;
- Macro No. 3620502 for the finishing-cycle.

Tool 31 is selected and a drilling cycle defined by the function G81. CNC control is transferred to UPC macro no. 3620501 by the use of E-parameter no.1 and the G22-function. The first starting points are drilled.

Blocks N10 to N16 command the coordinates of the starting-points macro to be mirrored about the X and Y-axes and for the macro to be executed after each mirroring. The G22-function transfers CNC control to the macro and the G14-function commands the block N9 to be repeated once.

Tool 4 is selected after the pocket has been executed four times. Block N32 assigns the value of 3620500 to E-parameter no.1; block N33 commands that blocks N10 to N16 are repeated. The 'machining' macro is therefore executed once and then mirrored and repeated in the same manner as for the 'starting-points' macro.

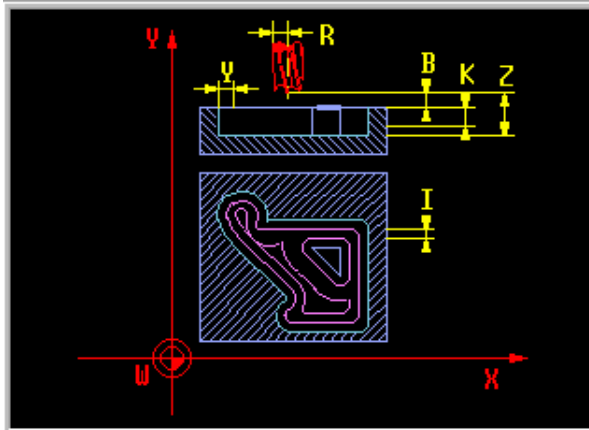
Tool 3 is selected. Block N36 assigns the value 3620502 to E-parameter no.1. The 'finishing' cycle is executed in the same manner as for the two previous UPC macros.

Block N38 commands the tool to move to Z100 and the partprogram to end.

5.66.2 G201 Start contour pocket cycle

Purpose

Entering technological data to calculate the pocket cycle. Pocket milling starts with this block.



Format

N... G201 N1=.. Z.. {N2=..} {Y..} {B..} {R..} {I..} {J..} {K..} {F..} {F2..}

Parameter

```
G      Start contour pocket cycle
Y      Stock removal
Z      Total pocket depth
B      Clearance
I      Cutting width mill in %
J      J1:climb / J-1:conventional
K      Cutting depth
R      Tool radius for calculation
N1=    Milling macro number
N2=    Startpoint macro number
```

Associated functions

G200...G208

For general description about universal pocket cycles see G200.

Parameter

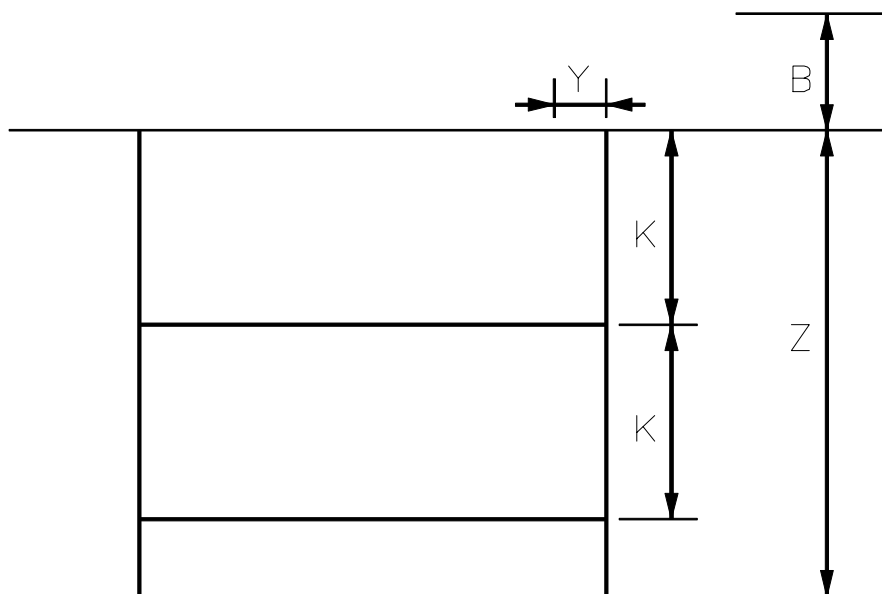
N1=: Number of the machining macro. This number has to be programmed.

N2=: Number of the starting points macro.

Y: Machining allowance = amount of material required to be left on the contour for finishing. The Y-word carries no sign. If Y has not been programmed, Y=0 will be used as a default.

Z: The total depth of the pocket. The depth is measured from the coordinate of the tool axis from the G203-block (which is equal to the position of the upper surface of the pocket).
 A negative Z-word value is the depth in the negative direction of the tool axis. A positive Z-word value is the depth in the positive direction of the tool axis. The Z-word must always be present in a block, which contains the G201-function. These words are independent of the selected machining plane.

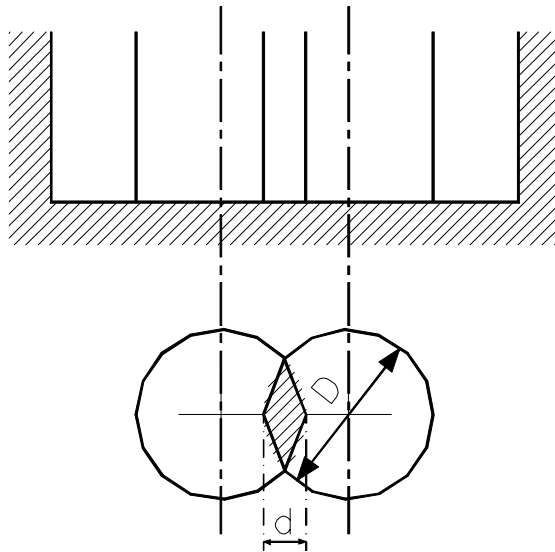
B : Clearance distance above the pocket. This distance is measured from the coordinate of the tool axis specified with the G203-function.
 Positive B-word value is the clearance distance in the positive direction of the tool axis; a negative value is the clearance distance in the negative direction. The sign of the B-word must always be opposite to the sign of the Z-word. If a B-word is not been programmed, B=0 will be used as a default.
 The B-word is used to define the position on the tool axis where the feed movement begins for entering the pocket at each starting point. At the end of the machining operation the tool is retracted above the pocket surface, at a distance stated by the B- Word.



R : The cutter radius to be used for calculating the cutter paths. The final cutter radius, which is actually used, can be different. The R-word carries no sign and must always be programmed.

I : The amount of overlap between cutting passes. The overlap distance is specified as a percentage of the tool's diameter e.g. I75= 75% of the diameter. I lies between 1% and 100%.

The I-word carries no sign. If the I-word is not programmed, the value of MC720 will be taken.



D = Cylinder cutter diameter
 d = overlap distance
 $I = 100 (1 - (d/D))$

- J** : The direction of movement for rough milling.
 J1 (default value): counter-clockwise direction
 J-1: clockwise direction.
- K** : The cutting depth per path. The K-word carries no sign. The last feed-in distance may be smaller than the value of K for the final cutting pass. If the value of K is larger than that of Z, or K-word is not stated, the Z-value will be used i.e. the value of $K = |Z|$.
- F** : The feed during milling.
 If F is not programmed, the last programmed feed will be used.
- F2=** : The feed for moving to a next machining plane. If the holes have been pre-drilled a great value can be taken. If F2= is not programmed, the last programmed feed will be taken.

The functions G90, G40 and G63 will be automatically activated when the G201-function is executed. G90 is required to be active because the generated macros use absolute dimensions and the first position of the pocket contour definition (G203-block) must be absolute.

5.66.2.1 Usage of the generated macros

STARTING POINT MACRO

The control is capable of generating a starting points macro. The starting points macro contains the points where the cutting tool enters the material for "roughing out" the pocket (these points are calculated by the control). The programmer can command a hole to be drilled in these positions, so that the cutting tool need not cut in the direction of the tool axis.

The macro generated by the CNC will be similar to that given below:

N (is N2- word of the G201-block)

N1 G90 Absolute programming

N2 G79 X.. Y.. Z.. Activate a predefined drilling cycle
 N3 G79 X.. Y.. Z..
 N4 G79 X.. Y.. Z.. Depending of number of drilling holes

The position in the machining plane XY (G17), XZ (G18) or YZ (G19) is given in the coordinates of the axes system in which the pocket has been described. The tool axis Z (G17), Y (G18) or X (G19) is given by the G203-block of the pocket contour.

After execution of the starting-points macro, G90 becomes active. The starting-points macro is generated simultaneously with the machining macro.

A partprogram could be as follows:

```
N9900
N1 G200
:
N90 T1 M6 (Drill)
N100 G81 X.. Y.. Z.. B.. F.. S.. M3
N110 G22 N=9902
:
N200 G201 Z.. N1=9901 N2=9902
: \
: > Description of the pocket (including islands)
: /
N300 G202
```

The cycle for pre-drilling the starting points is defined in block N100. The starting points macro is called in block N110. The cycle from block N100 is executed on the starting points.

MACHINING MACRO

The machining macro is generated by the control and includes all the movements necessary for roughing out a pocket. This macro (subprogram) will be called when the G201-function is executed. When the G202-function terminates the macro, the G40 and G90-functions will always become active automatically.

The example below shows a simple program, which uses a macro, which specifies a pocket cycle.

```
N (is N1-word in G201-block)
N1 G40      No radius correction
N2 G90      Absolute coordinates are used
N3 G0 X.. Y.. Z..      Tool moves to start point
N4 G91      Incremental coordinates are used
N5 G1 Z.. F..      Tool is fed to depth
N6 G90      Absolute coordinates are used
N7 F..      Feed
: \
: > Pocket milled
: /
N99
N100 G91
N101 G0 Z..      Tool is retracted
N102 G90      Absolute coordinates used again
N103 G0 X.. Y..      Tool moves back to starting point
N104 G14 N1=4 N2=103 J..      Pocket milling is repeated
N105 G91      Incremental coordinates are used
N106 G1 Z.. F..
N107 G14 N1=6 N2=99      Final cutting pass is performed
N108 G0 Z..      Tool retracts out of work piece
```

5.66.2.2 Macro for finishing a pocket contour

The macro for finishing the pocket contour is generated by the control. The macro includes all the movements including a circular or linear feed-in and feed-out, necessary for finishing the pocket sides.

FEED-IN POINT

The feed-in point S is determined by the CNC. This point is always near the start position of the pocket and at an equivalent distance (d) to both first and last sides.

The CNC will calculate different feed-in points depending on which contour side the tool has to first move to. The programmer therefore has to determine the correct tool size and the order in which the contour sides are stated, to prevent the tool colliding with the workpiece.

The CNC uses the formula below to calculate point S.

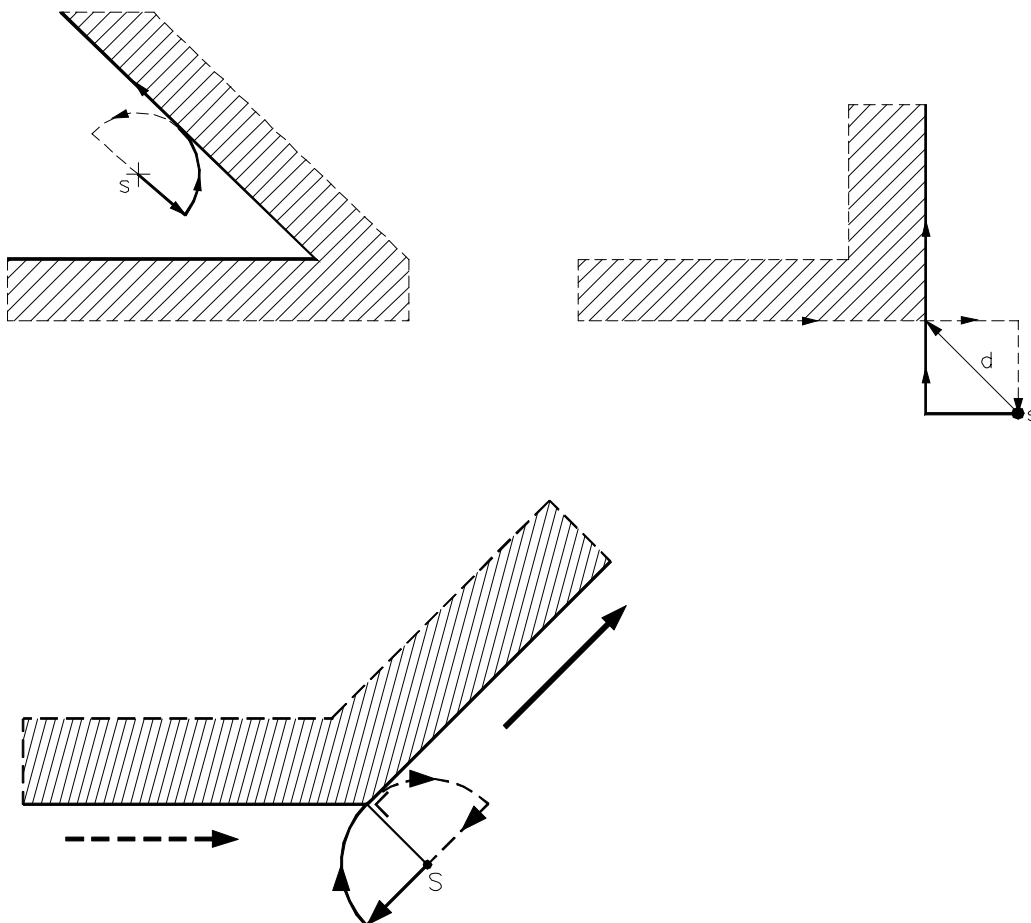
Distance = factor * radius : 100

factor = given per machine constant (MC 719) (between 101..200).

radius = the value of the R-word from the G201-block, or the active radius during G201, if not programmed

Within the circle, with S as centre and distance as radius no contour element of an island or pocket is allowed.

If it is not possible to calculate point S, the CNC generates an error code.

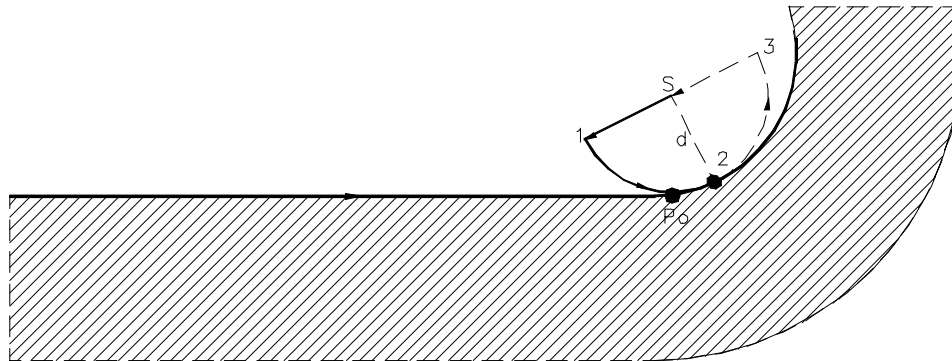


The feed-in movement is linear when the first and last movements are also linear and the angle between $\Rightarrow$ 270 degrees (this angle is measured inside the pocket). A circular feed-in movement is used in all other instances.

If a different tool is used for the finishing cut the programmer must ensure that the tool does not collide with the contour.

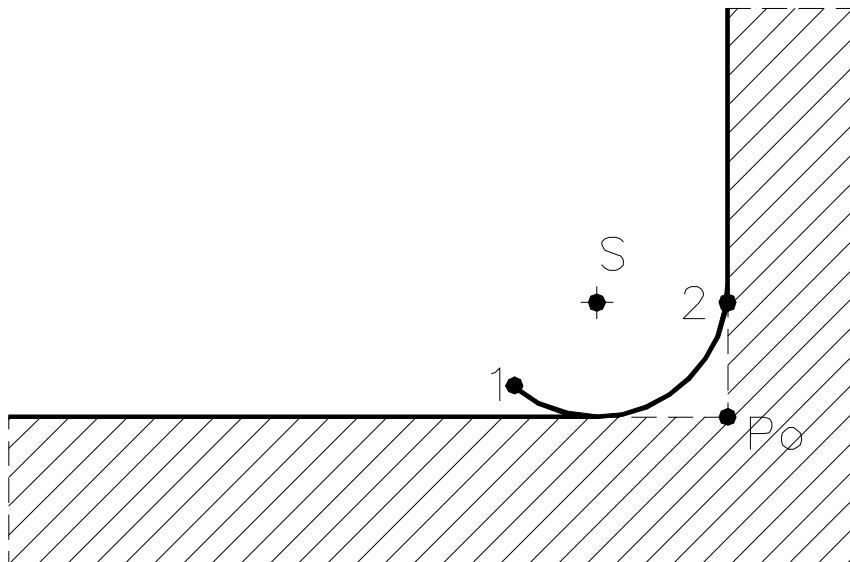
GENERAL FORMAT OF THE MACRO FOR A CIRCULAR FEED-IN

The feed-in of the contour is effected via a quarter-circle with centre S and a radius equal to the distance between S and the contour.



$$\text{distance} = \text{factor} * \text{radius} : 100$$

A problem will arise when the start point of the feed-in circular movement is on the path of the final contour. The tool will then slightly touch the contour surface. To prevent this occurring the CNC automatically repositions the start point S.



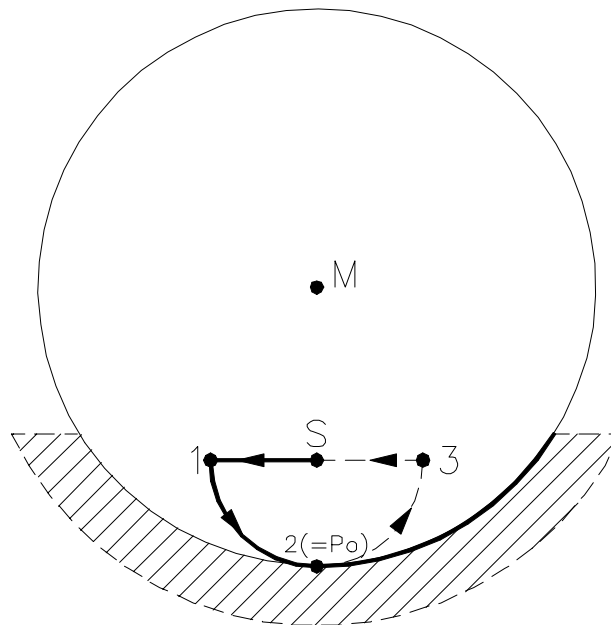
In this example the feed-in circle begins 120° before point 2.

A pocket wall can usually be machined in one step. If this is not possible, the programmer can finish the wall in several steps using datum point shifts.

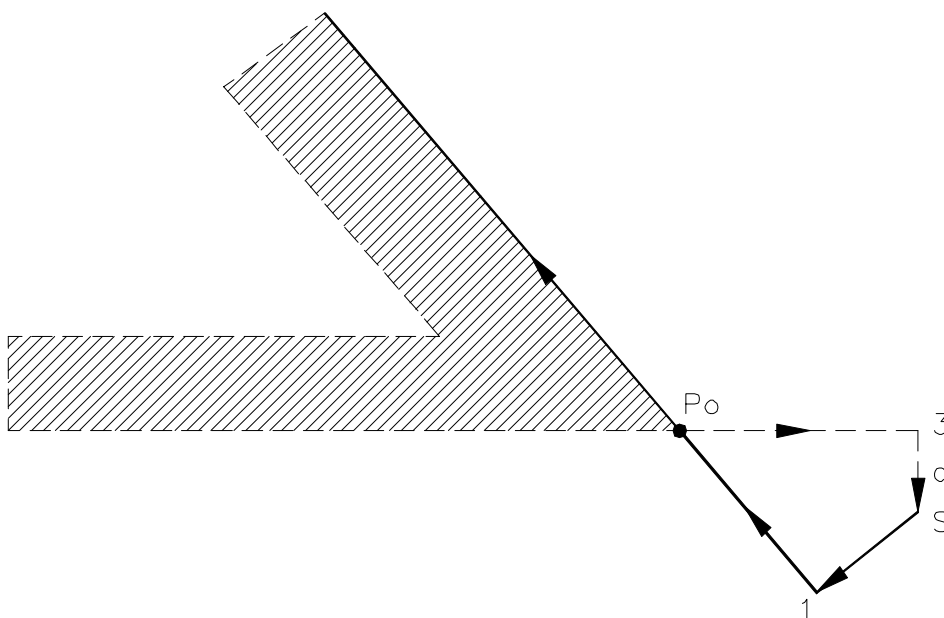
After completion of a macro, the functions G0, G40 and G90 become active.

CIRCULAR POCKET CONTOUR

For a circular pocket contour the calculation of the feed-in point S must be changed.



In this case point S lies on the straight line M to Po, at a calculated distance from Po. A pocket contour will be entered at point Po (= point 2). The direction of rotation at feed-in and feed-out is the same as the programmed direction of rotation for the contour.

LINEAR FEED-IN MOVEMENT

$$\text{distance} = \text{factor} * \text{radius} : 100$$

Note: After the macro is executed, the functions G0, G40 and G90 will be active.

5.66.2.3 Sequence of the macros on the machine

1. Starting points macro

This macro will run in the normal way without any restrictions.

2. Machining macro

This macro will run in the normal way as well, but some material may be left.

There are two possibilities:

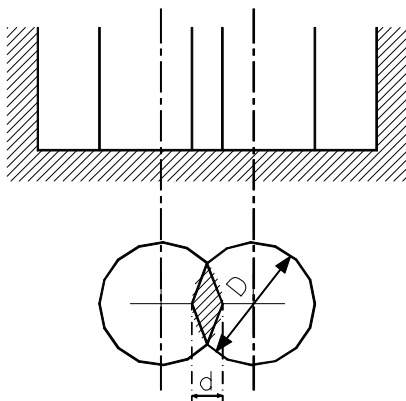
- (i). During contour-parallel milling small areas may remain unmilled when the tool changes direction between two parallel paths; this situation can also occur at contour roundings:

The solutions to these two problems are:

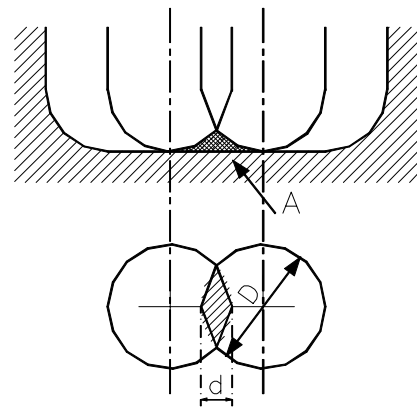
- a. For parallel movements - a bigger overlap between parallel movements.
- b. For contour narrowing - a tool with a smaller diameter is used.

- (ii) Due to the shape of the cutting tool some material may be left on the bottom. Normally, the overlap-parameter (I-word from the G201-block) should ensure that no material is left between the paths. The greater the value of the I-address, the greater the risk that material will be left on the bottom.

A value of 50% will ensure that when a square-edge-milling tool is used no material will be left in the pocket's bottom surface. However, milling tools with curved edges require different I-values because of the different radii of the curved edges.



Cylinder cutter



Ball cutter

D= cutter diameter
d= overlap distance
A= uncut material

3. Macros for finishing

The radius of the active tool will be used for radius compensation. The user should check that the cutting tool could move without damaging the pocket side. The macros will operate in the normal way.

Notes and usage

The function G201 and the terminating function G202 must both be written in the same program or in the same macro.

Between G201 and G202 only the following functions are permitted:

G203, G204, G205, G206 with contour descriptions
G207 and G208

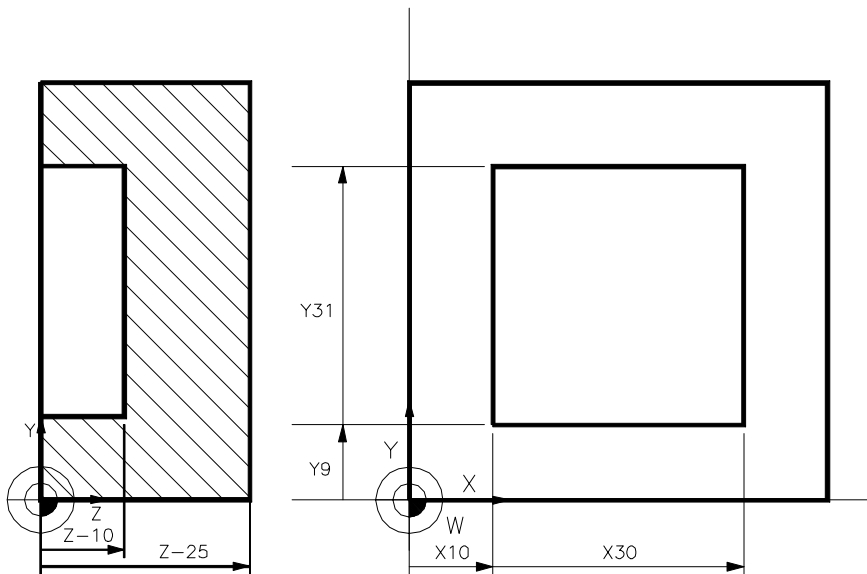
During description between G201 and G202 the following functions may not be used:

- Datum points shifts, notably (G92/G93) and stored zero offsets (G51 to G59) G54I[nr.]
- Axis rotation (G92, G93)
- Mirror image or scaling (G72, G73)
- Definition and activation of fixed cycles
- Measuring cycles
- Plane selection (G17, G18, G19)
- Point definition (G78)
- Helix interpolation
- Tool change (M06, M66, M67)
- Macro (G22) or program call (G23)
- Block and pattern repeat (G14)
- Conditional jump (G29)
- Chamfer or rounding (G11)

The use of E-parameters is not permitted for contour descriptions, or in the G201-block.

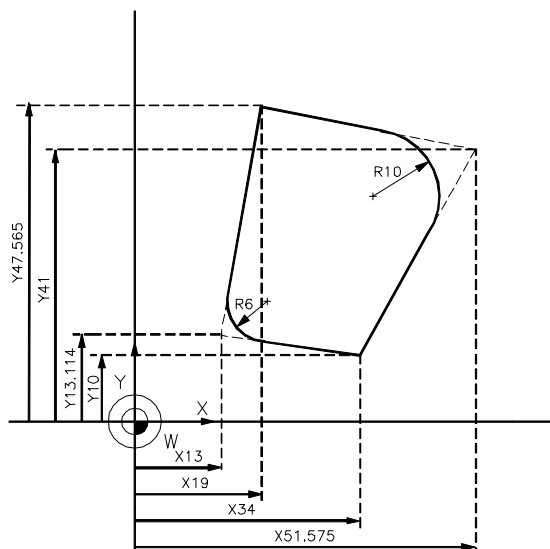
Example

Note: For the following examples it is assumed that G17 is active.

EXAMPLE 1 Rectangle

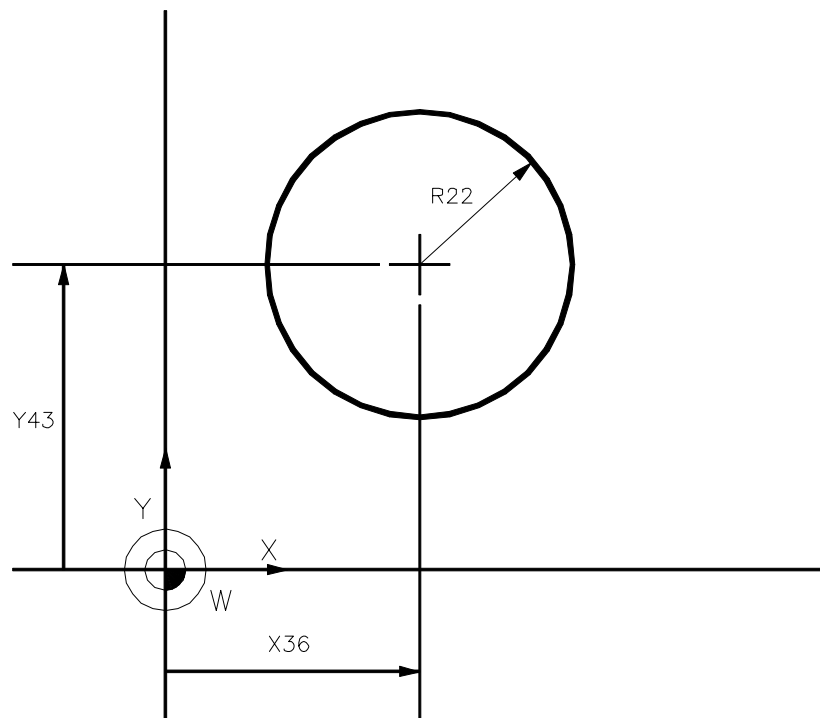
Programming can be done in two ways:

N.. G201 N1=.. Z-10	
N.. G203 X10 Y9 Z0 N1=9800	
Absolute	incremental
N.. X40	N.. G91
	N.. X30
N.. Y40	N.. Y31
N.. X10	N.. X-30
N.. Y9	N.. Y-31
N.. G204	
N.. G202	

EXAMPLE 2. Quadrangle with roundings

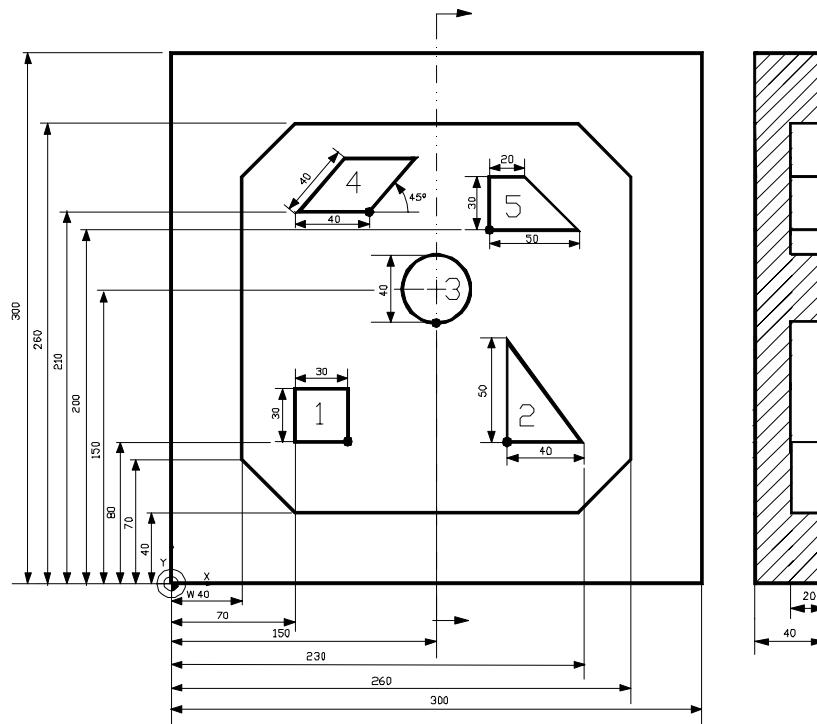
N.. G201 N1=.. Z-10
 N.. G203 X34 Y10 Z0
 N.. G64
 N.. G1 X51.575 Y41
 N.. G3 R10
 N.. G1 X19 Y47.565
 N.. G1 X1=13 Y1=13.114
 N.. G3 R6
 N.. G1 X34 Y10
 N.. G63
 N.. G204
 N.. G202

EXAMPLE 3. Full circle



N.. G201 N1=.. Z-10
 N.. G203 X36 Y21 Z0
 N.. G3 I36 J43
 N.. G204
 N.. G202

EXAMPLE 4. Pocket with islands

**N9990**

N1 G54

N2 G17

N3 G195 X-10 Y-10 Z10 I320 J320 K-60

N4 G99 X0 Y0 Z0 I300 J300 K-40

N5 G200

N6 T2 M6 (predrilling start point, drill R10)

N7 G81 Y2 Z-20 F200 S3000 M3

N8 G22 N=9992

N9 T3 M6 (clearing out the pocket, mill R8)

N10 S2500 M3

N11 G201 Y0.1 Z-20 B2 I50 K8 F200 F2=1000 N1=9991 N2=9992

:

N37 G202

5.66.3 G202 End contour pocket cycle

Purpose

End contour pocket cycle

Format

N... G202

Parameter

Non

Associated functions

G200...G208

For general description about universal pocket cycles see G200.

Notes and Usage

The function G201 and the terminating function G202 must both be written in the same program or in the same macro.

Between G201 and G202 only the following functions are permitted:

G203, G204, G205, G206 with contour descriptions
G207 and G208

During description between G201 and G202, the following functions may not be used:

- Datum points shifts, notably (G92/G93) and stored zero offsets (G51 to G59) (G54I[nr.])
- Axis rotation (G92, G93)
- Mirror image or scaling (G72, G73)
- Definition and activation of fixed cycles
- Measuring cycles
- Plane selection (G17, G18, G19)
- Point definition (G78)
- Helix interpolation
- Tool change (M06, M66, M67)
- Macro (G22) or program call (G23)
- Block and pattern repeat (G14)
- Conditional jump (G29)
- Chamfer or rounding (G11)

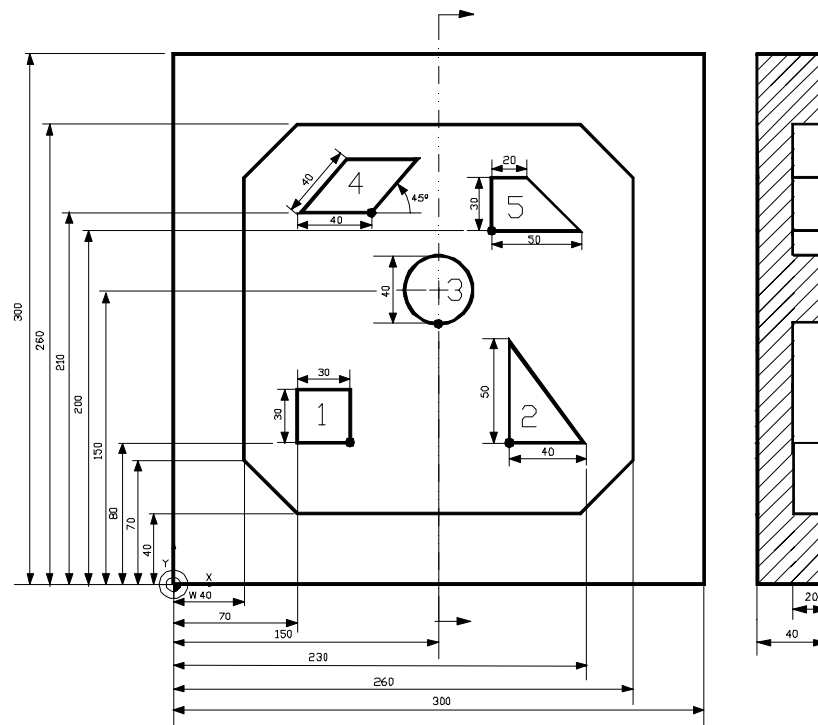
Completion of the entire pocket description. After the pocket has been cut out, the partprogram will be resumed with the block following G202. Only the N-word is permitted in a G202-block.

By a G202-function the calculation for the universal pocket cycle will be stopped. After a new G200-function the calculation will be started again.

The functions G0, G40, G63 and G90 will be active after the G202-function is activated.

After the pocket description the program should be continued with an absolute position.

EXAMPLE: Pocket with islands

**N9990**

N1 G54

N2 G17

N3 G195 X-10 Y-10 Z10 I320 J320 K-60

N4 G99 X0 Y0 Z0 I300 J300 K-40

N5 G200

N6 T2 M6 (predrilling start point, drill R10)

N7 G81 Y2 Z-20 F200 S3000 M3

N8 G22 N=9992

N9 T3 M6 (clearing out the pocket, mill R8)

N10 S2500 M3

N11 G201 Y0.1 Z-20 B2 I50 K8 F200 F2=100 N1=9991 N2=9992

N12 G203 X40 Y40 Z0 N1=9993

N13 G208 X220 Y220 I30 (pocket contour)

N14 G204

N15 G205 X100 Y80 N1=9994

N16 G208 X-30 Y30 J-1 (Island 1)

N17 G206

N18 G205 X190 Y80 N1=9995

N19 G91

N20 Y50 (Island 2)

N21 X40 Y-50

N22 G90

N23 G206

N24 G205 X150 Y130 N1=9996

N25 G2 I150 J150 (Island 3)

N26 G206

N27 G205 X110 Y210 N1=9997

N28 G208 X-40 Y40 J-1 B1=135 (Island 4)

N29 G206

N30 G205 X180 Y200 N1=9998

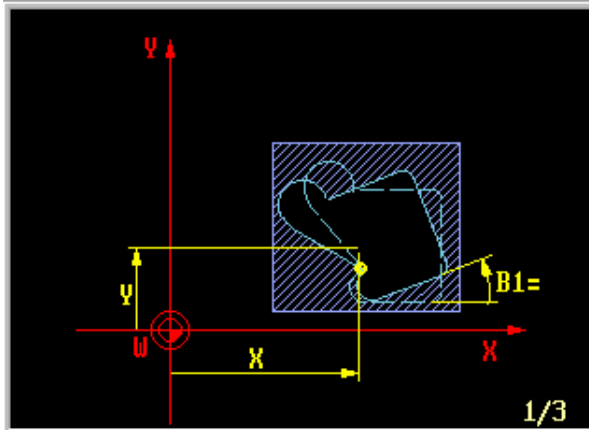
N31 G91

N32 Y30
N33 X20 (Island 5)
N34 X30 Y-30
N35 G90
N36 G206
N37 G202
N38 T4 M6 (clearing out the pocket, mill R8)
N39 F200 S2200 M3 |
N40 G22 N=9993 |
N41 G22 N=9994 |
N42 G22 N=9995 | (finishing)
N43 G22 N=9996 |
N44 G22 N=9997 |
N45 G22 N=9998 |
N46 M30

5.66.4 G203 Start pocket contour description

Purpose

Start pocket contour description



Format

N... G203 X.. Y.. Z.. {B1=..} {N1=..}

Parameter

```
G   Start pocket contour description
X   Startpoint in X
Y   Startpoint in Y
Z   Startpoint in Z
P   Point definition number
N1= Finishing macro number
B1= Rotation angle of contour pocket
B2= Start point polar angle
L2= Start point polar length
P1= Point definition number
```

N1=: Number of the macro for finishing the pocket contour. A finishing macro will not be generated if the word N1=.. is not programmed.

B1=: Rotation of the pocket contour around the point from the G203-block. Islands will not be rotated.

Associated functions

G200...G208

For general description about universal pocket cycles see G200.

Notes and Usage

The description of the pocket contour is started by stating the position of the first point: the finishing cut of the contour also starts at this point.

The starting point can be defined by absolute Cartesian coordinates measured from the program zero point W. The starting point can also be programmed by using absolute polar coordinates (B2=... and L2=...). The position can also be programmed by a predefined (G78) point.

The position of the workpiece's upper surface is defined by coordinate values of the following tool axes: (G17) Z-axis, (G18) X-axis, (G19) and Y-axis.

A tool axis coordinate must always be present in a G203-function's block.

G203 causes G1, G63 and G90 to become active.

The contour description can start at any point, e.g. the middle of a side's length.

The sides of the contour can be described using all the possibilities of the control to indicate linear and circular sides.

The pocket contour must be closed; if a gap is present the CNC will automatically close that gap with a linear contour side.

Absolute or incremental coordinates are allowed in the contour description. Programming is effected via G90 or G91.

The geometric function (G64) may be used. The same conditions will apply as for programming a geometric contour. The most important is that between G64 and G63 absolute coordinates must be used exclusively.

Only axes in the main plane may be programmed. The program can start its finishing cut from the middle of a side's length; this is done by positioning the start point of the pocket cycle description at the middle of the side.

The program starts its finishing cut at the start point of the contour description. The finishing cycle will follow the same sequence as the contour description.

Only the following functions are permitted between G203 and G204, or between G205 and G206:

G1, G2, G3, G208 G63, G64 G90, G91

The G1, G2/G3 movements are limited to the main plane. Tool-axis and rotary axis coordinates are not permitted.

The G63/G64 and G90/G91 functions must all be programmed in separate blocks e.g.:

N10 G64

N20 X... Y... : This is acceptable.

N10 G64 X... Y... : This is not acceptable.

The associated pair of functions G203/G204 and G205/G206 must be written in the same program or the same macro.

The first point of a contour description must be given in a G203-block.

Defined points may be used in a contour, but a point definition (G78) is not allowed. The points must have been defined prior to the G200-block of the pocket.

The bottom of the pocket must be parallel to the machining plane, i.e. XY-plane (G17), XZ-plane (G18) or YZ-plane (G19). A slanted or a curved bottom plane is not allowed.

The sides of the pocket must be perpendicular to the bottom plane.

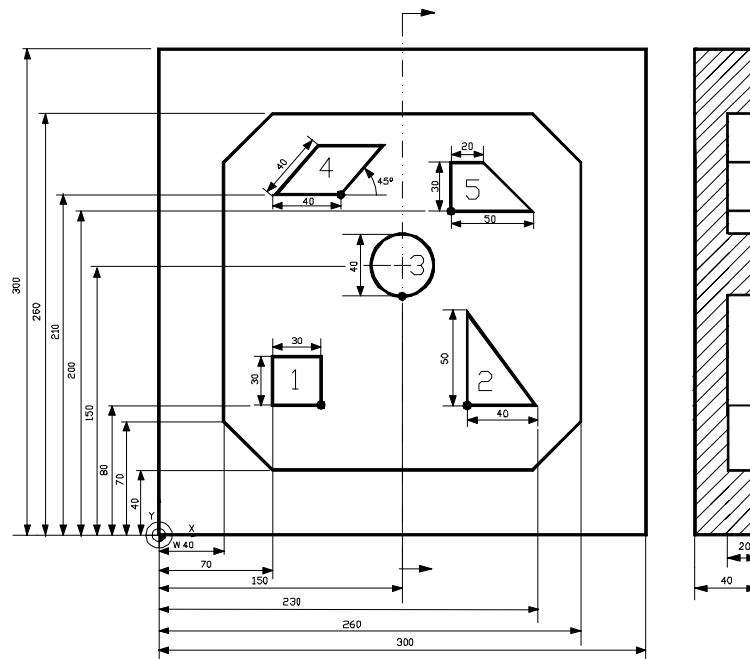
Two parts of the same pocket may not intersect, or be tangent, to each other. However, two different pockets may intersect each other.

During finishing the programmer should make sure that the tool diameter is selected smaller than the distance to the narrowest part in pocket of the workpiece. The control system is unable to identify contours damaged during finishing.

The rules, which apply to G1 and G2/G3 movements, also apply to the same types of movement used within pocket cycles.

When the (G64) Geometry function is active, only absolute (G90) coordinates can be used and only one predefined (G78) point can be present in a program block.

EXAMPLE Pocket with islands



N9990

N1 G54

N2 G17

N3 G195 X-10 Y-10 Z10 I320 J320 K-60

N4 G99 X0 Y0 Z0 I300 J300 K-40

N5 G200

N6 T2 M6 (predrilling start point, drill R10)

N7 G81 Y2 Z-20 F200 S3000 M3

N8 G22 N=9992

N9 T3 M6 (clearing out the pocket, mill R8)

N10 S2500 M3

N11 G201 Y0.1 Z-20 B2 I50 K8 F200 F2=100 N1=9991 N2=9992

N12 G203 X40 Y40 Z0 N1=9993

:

N14 G204

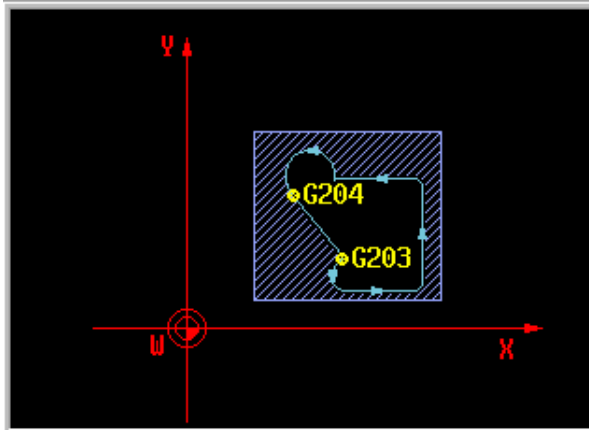
:

N37 G202

5.66.5 G204 End pocket contour description

Purpose

End pocket contour description



Format

N... G204

Parameter

Non

Associated functions

G200...G208

For general description about universal pocket cycles see G200.

Notes and Usage

G204 is only allowed to be programmed between G201 and G202.

Only the following functions are permitted between G203 and G204:

G1, G2, G3, G208, G63, G64, G90, G91

The G1, G2/G3 movements are limited to the main plane. Tool-axis and rotary axis coordinates are not permitted.

The G63/G64 and G90/G91 functions must all be programmed in separate blocks e.g.:

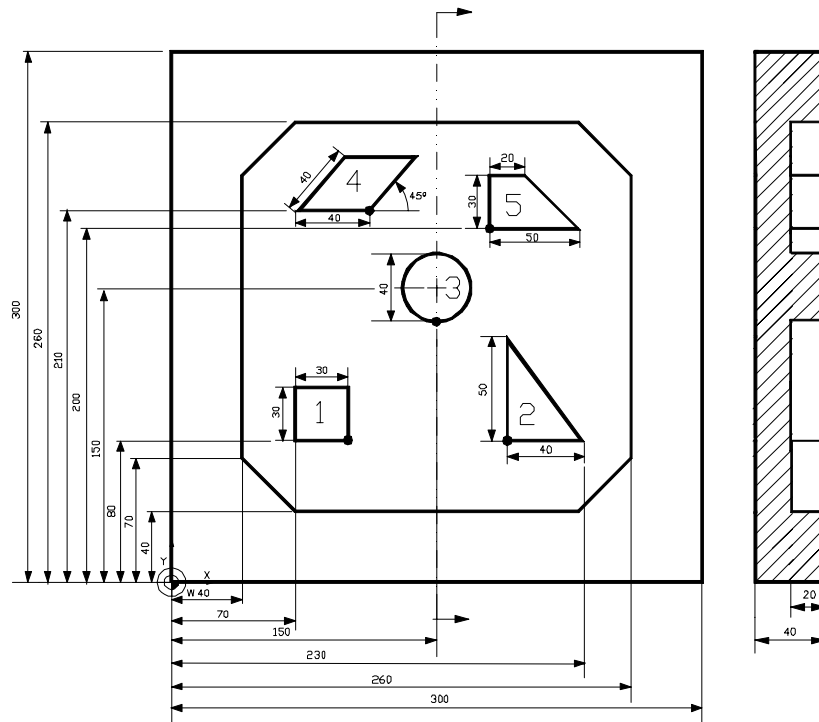
N10 G64

N20 X... Y... : This is acceptable.

N10 G64 X... Y... : This is not acceptable.

The associated pair of functions G203/G204 must be written in the same program or the same macro.

EXAMPLE Pocket with islands

**N9990**

N1 G54

N2 G17

N3 G195 X-10 Y-10 Z10 I320 J320 K-60

N4 G99 X0 Y0 Z0 I300 J300 K-40

N5 G200

N6 T2 M6 (predrilling start point, drill R10)

N7 G81 Y2 Z-20 F200 S3000 M3

N8 G22 N=9992

N9 T3 M6 (clearing out the pocket, mill R8)

N10 S2500 M3

N11 G201 Y0.1 Z-20 B2 I50 K8 F200 F2=100 N1=9991 N2=9992

N12 G203 X40 Y40 Z0 N1=9993

N13 G208 X220 Y220 I30 (pocket contour)

N14 G204

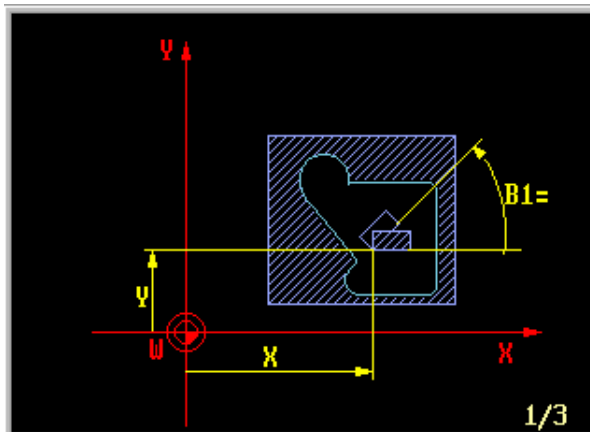
:

N37 G202

5.66.6 G205 Start island contour description

Purpose

Start island contour description



Format

N... G205 X.. Y.. {N1=...}

Parameter

```

G   Start island contour description
X   Startpoint in X
Y   Startpoint in Y
P   Point definition number
N1= Finishing macro number
B1= Rotation angle of island contour
B2= Start point polar angle
L2= Start point polar length
P1= Point definition number

```

N1= Number of the macro for finishing the pocket contour. A finishing macro will not be generated if the word N1=.. is not programmed.

B1= Rotation of the pocket contour around the point from the G203-block. Islands will not be rotated.

Associated functions

G200...G208

For general description about universal pocket cycles see G200.

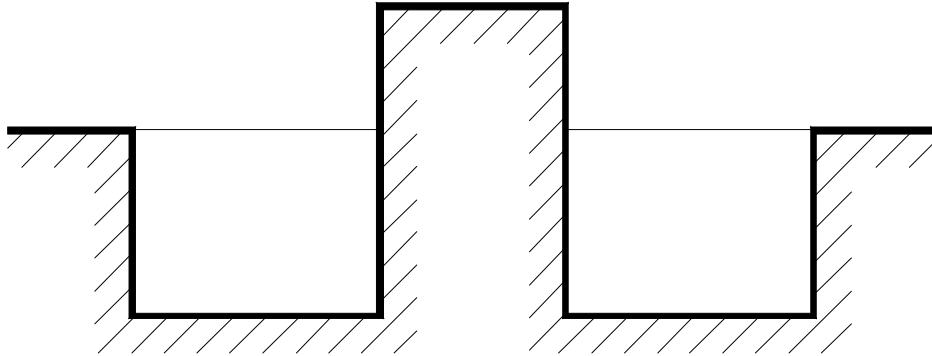
5.66.6.1 Description of the island contour

The contour of an island is described in the same way as the contour of a pocket. The description begins with G205 and the absolute starting position of the island.

The absolute position is described with either:

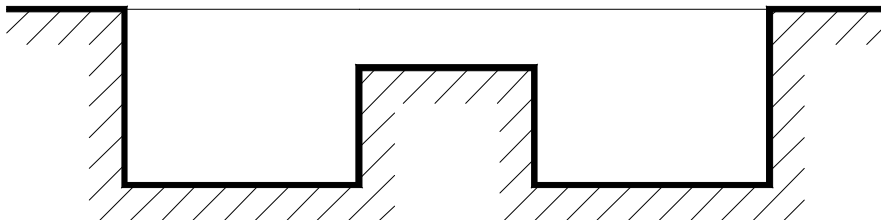
- Cartesian coordinates
- polar coordinates
- a defined point.

Programming of the tool axis is not allowed. The CNC assumes that the top surface of the island coincides with the top surface of the pocket.



If the island rises above the upper surface of the pocket, the B-word from the G201-block must be used to prevent a collision between cutting tool and workpiece during a movement from one starting point to the other.

If the upper surface of the island lies below the upper surface of the pocket, the partprogrammer must see to it that, after the pocket has been cut, the island is brought to the correct level.

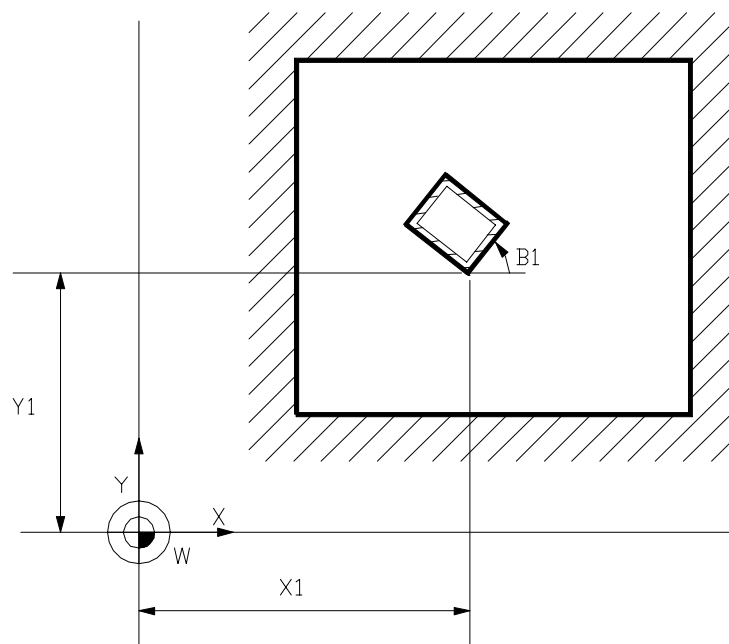


G205 causes G1, G63 and G90 to become active.

The contour description is terminated with G206.

For more information about island contour description see G203.

ROTATION OF AN ISLAND AROUND THE STARTING POINT



The word B1=.. from the G205-block indicates that the island is rotated around its starting point (X1, Y1) through the programmed angle.

The programming will be:

```
G205 X (=X1) Y (=Y1) B1= (=B1)    N1=..
: \
: >..Island contour
: /
G206
```

Notes and usage

Only the following functions are permitted between G205 and G206:

G1, G2, G3, G208 G63, G64 G90, G91

The G1, G2/G3 movements are limited to the main plane. Tool-axis and rotary axis coordinates are not permitted.

The G63/G64 and G90/G91 functions must all be programmed in separate blocks e.g.:

N10 G64

N20 X... Y... : This is acceptable.

N10 G64 X... Y... : This is not acceptable.

The associated pair of functions G205/G206 must be written in the same program or the same macro.

The contour of an island must be closed.

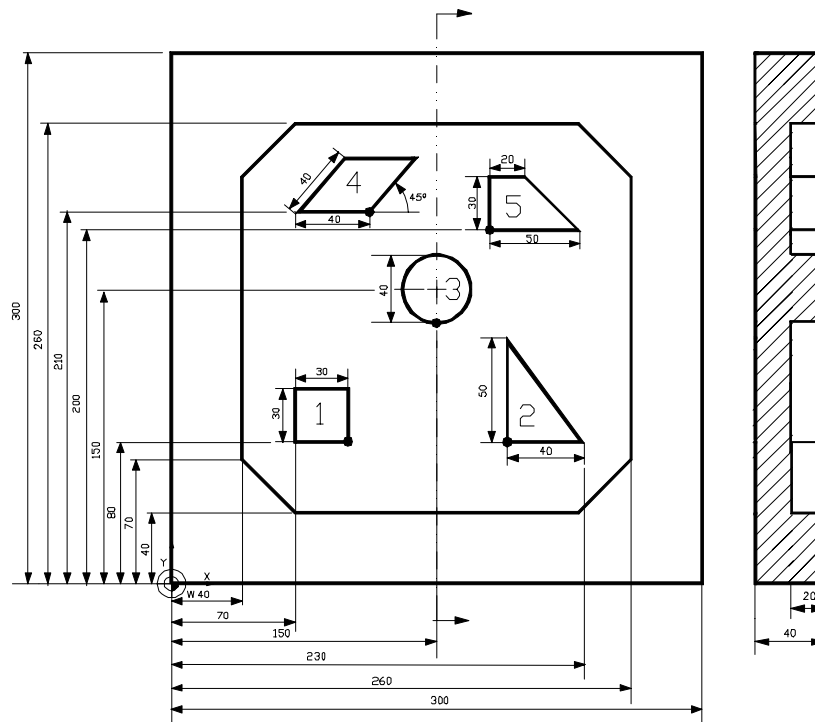
Two islands may not intersect with each other or be tangent.

Islands must be situated in the pocket and may not intersect with, or be tangent to, the pocket sides.

The sides of an island must be perpendicular to the bottom plane.

An island may not be enclosed by another island.

EXAMPLE Pocket with islands

**N9990**

N1 G54

N2 G17

N3 G195 X-10 Y-10 Z10 I320 J320 K-60

N4 G99 X0 Y0 Z0 I300 J300 K-40

N5 G200

N6 T2 M6 (predrilling start point, drill R10)

N7 G81 Y2 Z-20 F200 S3000 M3

N8 G22 N=9992

N9 T3 M6 (clearing out the pocket, mill R8)

N10 S2500 M3

N11 G201 Y0.1 Z-20 B2 I50 K8 F200 F2=100 N1=9991 N2=9992

N12 G203 X40 Y40 Z0 N1=9993

N13 G208 X220 Y220 I30 (pocket contour)

N14 G204

N15 G205 X100 Y80 N1=9994

N16 G208 X-30 Y30 J-1 (Island 1)

N17 G206

N18 G205 X190 Y80 N1=9995

N19 G91

N20 Y50 (Island 2)

N21 X40 Y-50

N22 G90

N23 G206

N24 G205 X150 Y130 N1=9996

N25 G2 I150 J150 (Island 3)

N26 G206

N27 G205 X110 Y210 N1=9997

N28 G208 X-40 Y40 J-1 B1=135 (Island 4)

N29 G206

N30 G205 X180 Y200 N1=9998

N31 G91

N32 Y30

N33 X20 (Island 5)

N34 X30 Y-30

N35 G90

N36 G206

N37 G202

N38 T4 M6 (clearing out the pocket, mill R8)

N39 F200 S2200 M3

N40 G22 N=9993

N41 G22 N=9994

N42 G22 N=9995 | (finishing)

N43 G22 N=9996

N44 G22 N=9997

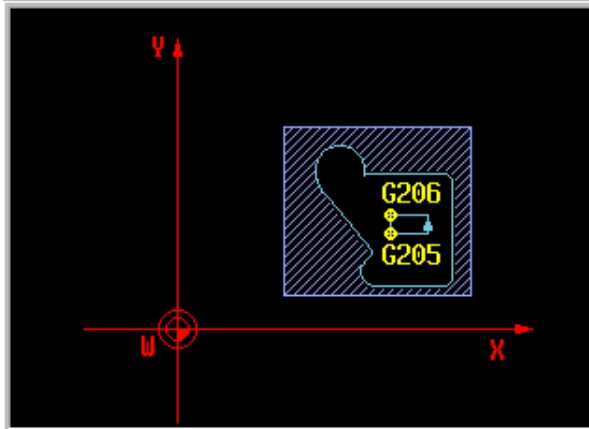
N45 G22 N=9998

N46 M30

5.66.7 G206 End pocket contour description

Purpose

End pocket contour description



Format

N... G206

Parameter

Non

Associated functions

G200...G208

For general description about universal pocket cycles see G200.

Notes and Usage

Only the following functions are permitted between G205 and G206:

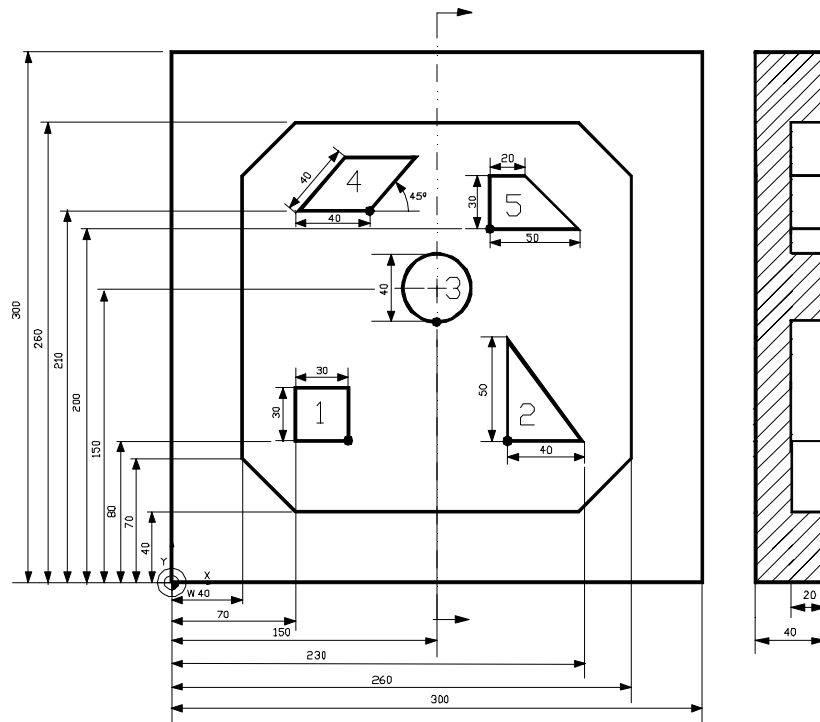
G1, G2, G3, G208

G63, G64

G90, G91

The associated pair of functions G205/G206 must be written in the same program or the same macro.

EXAMPLE Pocket with islands

**N9990**

N1 G54

N2 G17

N3 G195 X-10 Y-10 Z10 I320 J320 K-60

N4 G99 X0 Y0 Z0 I300 J300 K-40

N5 G200

N6 T2 M6 (predrilling start point, drill R10)

N7 G81 Y2 Z-20 F200 S3000 M3

N8 G22 N=9992

N9 T3 M6 (clearing out the pocket, mill R8)

N10 S2500 M3

N11 G201 Y0.1 Z-20 B2 I50 K8 F200 F2=100 N1=9991 N2=9992

N12 G203 X40 Y40 Z0 N1=9993

N13 G208 X220 Y220 I30 (pocket contour)

N14 G204

N15 G205 X100 Y80 N1=9994

N16 G208 X-30 Y30 J-1 (Island 1)

N17 G206

N18 G205 X190 Y80 N1=9995

N19 G91

N20 Y50 (Island 2)

N21 X40 Y-50

N22 G90

N23 G206

N24 G205 X150 Y130 N1=9996

N25 G2 I150 J150 (Island 3)

N26 G206

N27 G205 X110 Y210 N1=9997
N28 G208 X-40 Y40 J-1 B1=135 (Island 4)

N29 G206

N30 G205 X180 Y200 N1=9998
N31 G91
N32 Y30
N33 X20 (Island 5)
N34 X30 Y-30
N35 G90

N36 G206

N37 G202
N38 T4 M6 (clearing out the pocket, mill R8)
N39 F200 S2200 M3 |
N40 G22 N=9993 |
N41 G22 N=9994 |
N42 G22 N=9995 | (finishing)
N43 G22 N=9996 |
N44 G22 N=9997 |
N45 G22 N=9998 |
N46 M30

5.66.8 G207 Call island contour macro

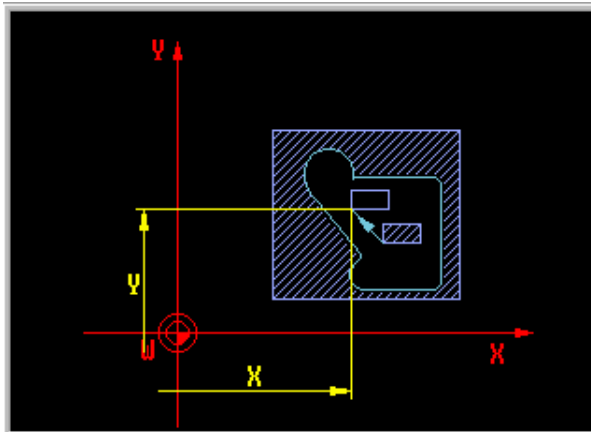
Purpose

Programs the same island contour in another place

Three possibilities are available:

1. The same island occurs in another place within the same pocket contour.
2. The same island contour occurs in another pocket contour.
3. The same island contour occurs in another program.

By including the island contour in a macro the three possibilities can be treated in the same way.



Format

N... G207 N= X... Y... {Z...} {N1=...}

Associated functions

G200...G208

For general description about universal pocket cycles see G200.

Parameter

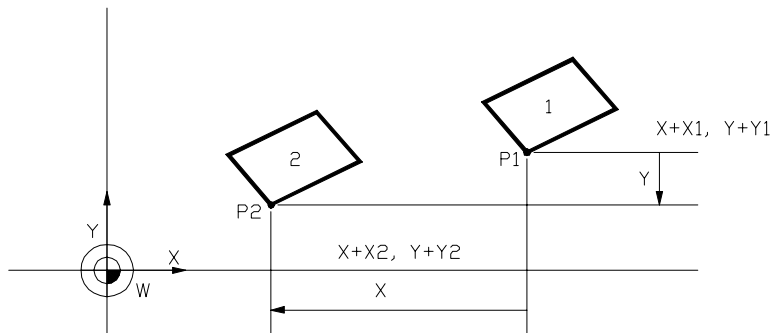
G	Call island contour macro
X	Shift along in X
Y	Shift along in Y
Z	Shift along in Z
N=	Macro with islandcontour
N1=	Finishing macro number

N=..	:	Macro number with island contour
X, Y		The distance between the start point in the macro of the programmed contour and the desired start point

Notes and Usage

Geometric programmed contours (G64...G63) within a G207 macro (pocket cycle) may function incorrect because the macro-offset is only taken into account at the start point. Programmers using pocket cycle are advised to check the pocket macros on this phenomenon and if so to incorporate the macro concerned into the partprogram in order to overcome the problem.

Example



The macro of the island contour will then be:

N9xxx G205 X=X2 Y=Y2 N1=.. (X and Y are the start coordinates of the island contour in respect to the workpiece zero point.

```
N1      \
: > Island contour
N..     /
N.. G206
```

Here, N9xxx is the identification of the macro.

The macro is called with the function G207.

```
N.. G201
```

```
N.. G207 N=9xxx N1=....
```

```
N.. G207 N=9xxx X=(X1-X2) Y=(Y1-Y2) N1=....
```

```
N.. G202
```

Explanation:

1: Island whose contour is programmed as a macro.

P1: Starting point of the contour description (G205-block).

2: Desired position of the island.

P2: Starting point of the shifted contour.

X...: Distance parallel to X-axis from P1 to P2.

Y...: Distance parallel to Y-axis from P1 to P2.

The distance carries a sign, as with incremental programming.

Note

The best way is too start the island contour with the coordinates X0, Y0. (Zero point shift in G205). In the G207 block the start point can programmed, without calculating.

The same macro of the island contour is than:

```
N9xxx G205 X0 Y0 N1=..  
N.. \  
: > Island contour with zero point shift  
N.. /  
N.. G206
```

N9xxx is the macro number.

The macro is called with the G207 function.

```
N.. G201  
:  
N.. G207 N=9xxx X=X2 Y=Y2  
N.. G207 N=9xxx X=X1 Y=Y1  
N.. G202
```

G63/G64 are not permitted in the macro.

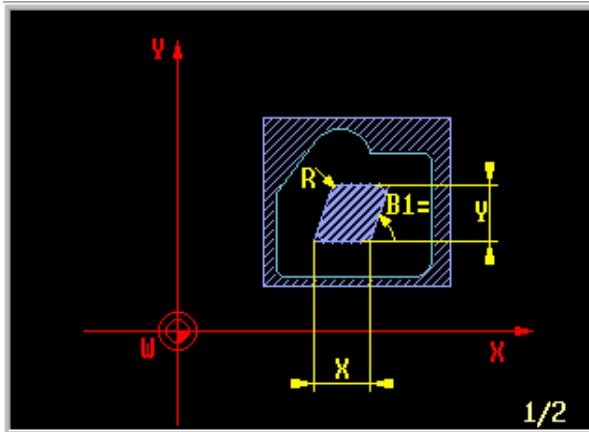
Absolute and/or incremental programming is possible.

5.66.9 G208 Quadrangle contour description

Purpose

To describe a regular rectangle as a pocket or island contour.

The G208 can be used to easily program a regular rectangle, more in particular a rectangle or parallelogram, as a contour.



Format

N... G208 X.. Y.. Z.. {I..} {J..} {R..} {B1=..}

Associated functions

G200...G208

For general description about universal pocket cycles see G200.

Parameter

G	Quadrangle contour description
X	Length in X
Y	Length in Y
Z	Length in Z
I	Chamfer length
J	J1:climb / J-1:conventional
R	Rounding radius
B1=	Angle quadrangle contour

N: Block number

X and Y: These words state distances along the two main-plane axes. These distances are measured from the start point stated by the G203-function. The sign of each word states the direction in which a distance was measured. A '+' distance is measured in a positive axis direction and a '-' distance measured in a negative axis direction.

J : The direction of the finishing movement will be determinate by the J-address:

J1 Milling in counter clockwise direction (climb milling)

J-1 Milling in clockwise direction (conventional milling)

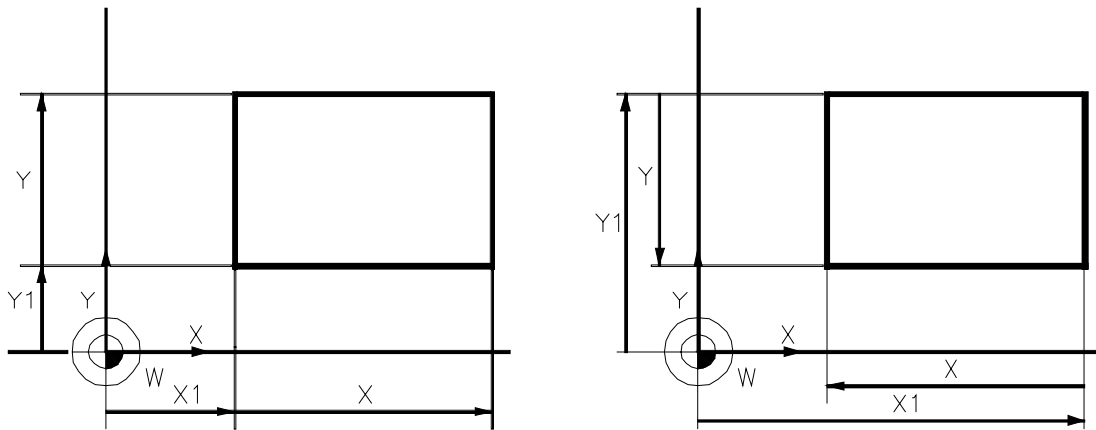
J is automatically 1, when no J-Address is programmed.

Notes and Usage

The bottom of the pocket must always be parallel to the main-plane.

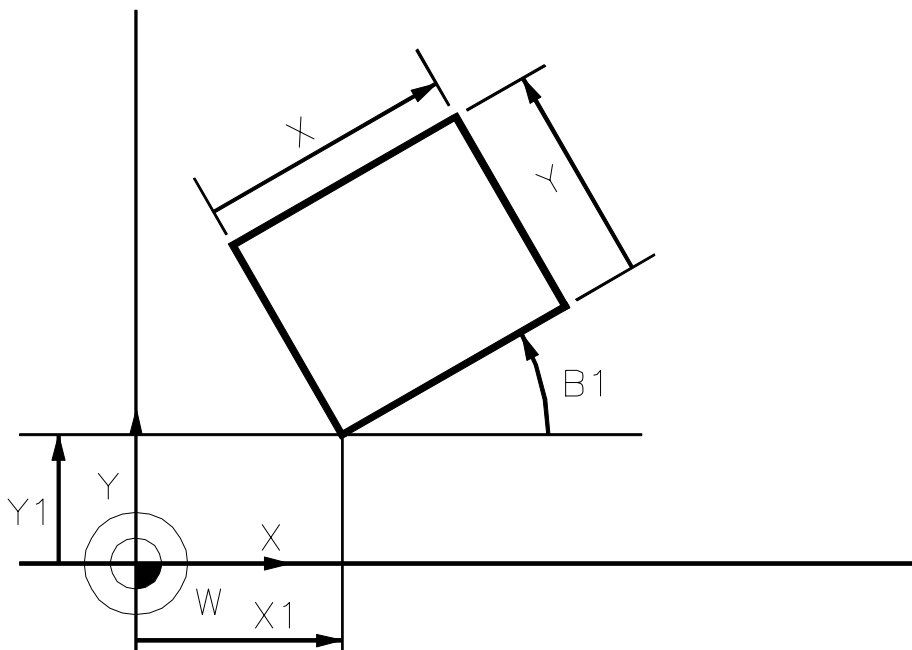
Examples

EXAMPLE 1 Rectangle



If the sides of a rectangle are parallel to the axes, the X and Y coordinates are equally parallel to the axes.

If the sides are not parallel to the X and Y-axis, G208 is programmed as if the sides are parallel. The G203- block contains the rotation angle (B1=...).



Programming will be:

G203 X (=X1) Y (=Y1) Z (=Z1) B1= (=B1)

G208 X (=X) Y (=Y)

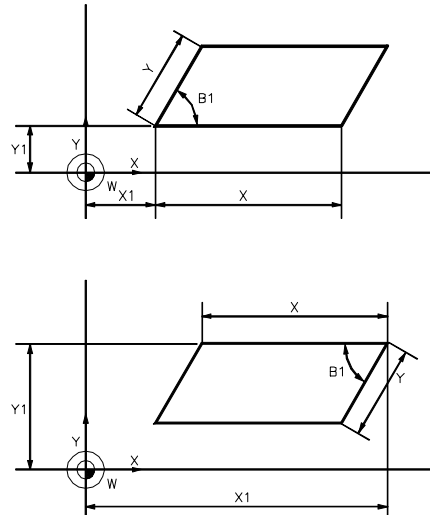
G204

EXAMPLE 2 Parallelogram

To program a parallelogram the lengths of both sides and the angle between them at the start point are stated.

Here:

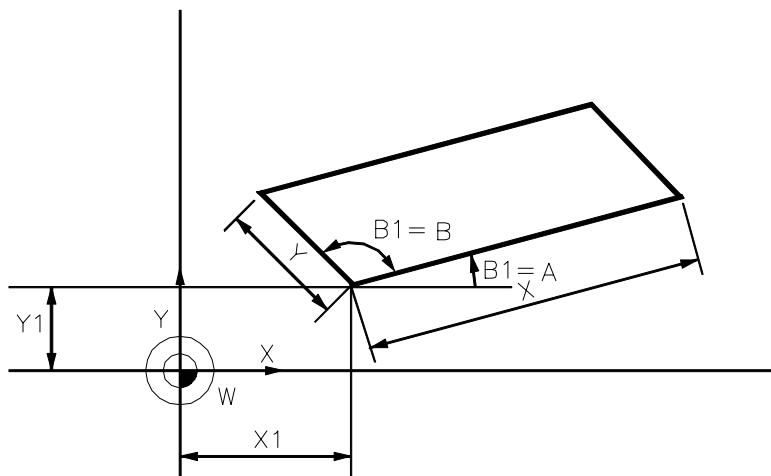
- B1=: The angle in degrees and decimal fractions of degrees ($0^\circ < B1 < 180^\circ$).
 The angle carries no sign.
 The default value for B1=.. is 90 degrees, i.e. a rectangle.



Programming will be:

```
G203 X (=X1) Y (=Y1) Z (=Z1)
G208 X (=X) Y (=Y) B1= (=B1)
G204
```

If one side is not parallel to the X-axis, programming will be the same. The G203-block contains the rotation angle.



Programming will be:

```
G203 X (=X1) Y (=Y1) Z (=Z1) B1= (=A)
G208 X (=X) Y (=Y) B1= (=B)
G204
```

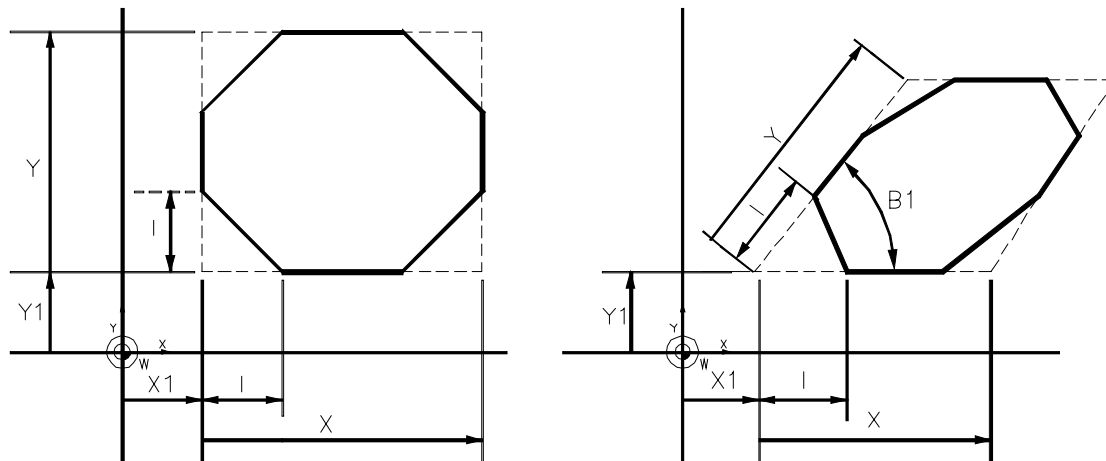
EXAMPLE 3 One chamfer

For both the rectangle and the parallelogram a bevel or phase can be added.

I-word: The width of the chamfer.

The I-word carries no sign.

The chamfer is symmetrically arranged around the corner.



Programming will be:

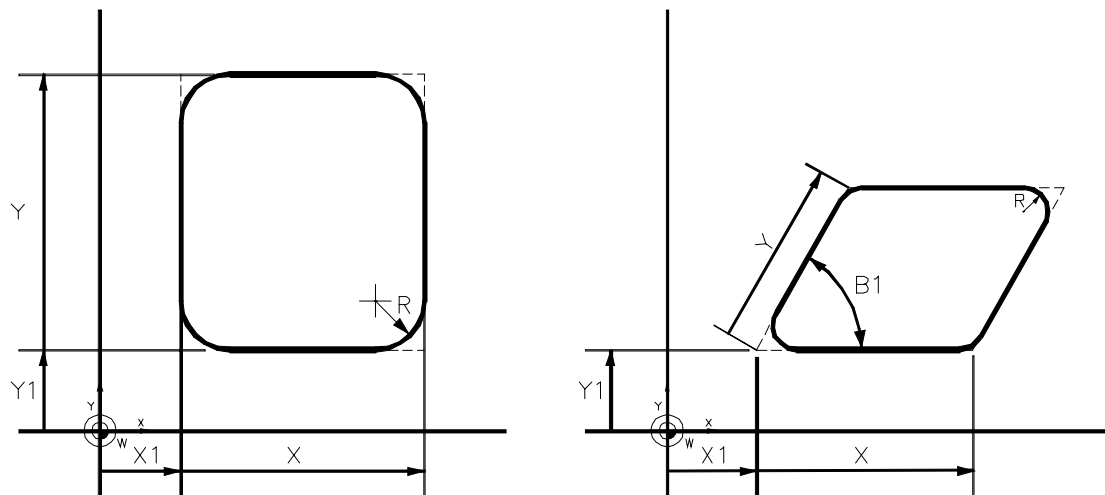
G203 X (=X1) Y (=Y1) Z (=Z1)

G208 X (=X) Y (=Y) B1= (=B1) I (=I)

G204

EXAMPLE 4 Rounding

R-word: The radius of the rounding. The radius does not have a '+/-' sign.



Programming will be:

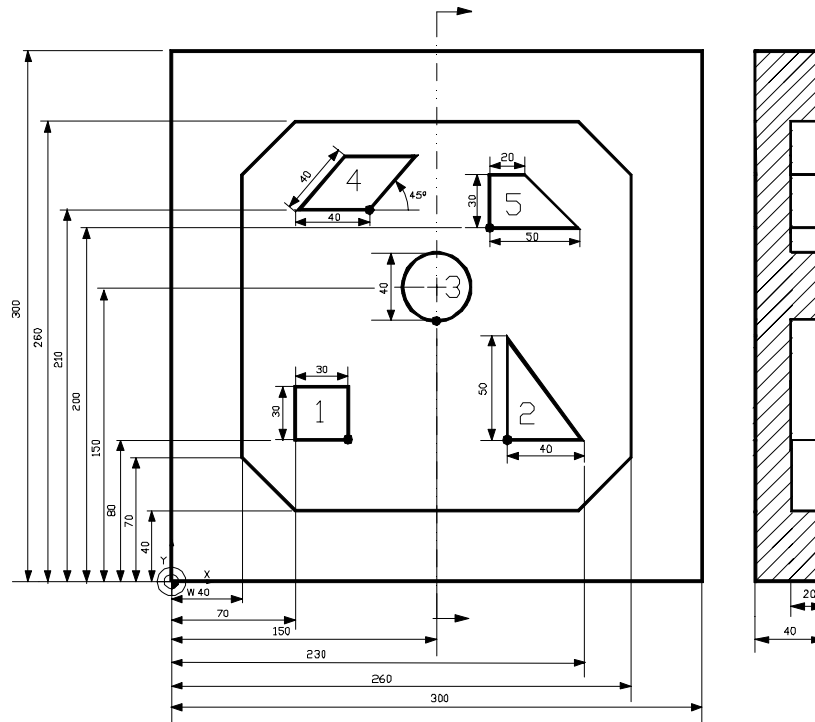
G203 X (=X1) Y (=Y1) Z (=Z1)

G208 X (=X) Y (=Y) B1= (=B1) R (=R)

G204

Either the I-word or the R-word can be programmed: the two words cannot be used at the same time, if this is done, an 'invalidation contour description' message (P144) is generated.

EXAMPLE 5 Pocket with islands

**N9990**

N1 G54

N2 G17

N3 G195 X-10 Y-10 Z10 I320 J320 K-60

N4 G99 X0 Y0 Z0 I300 J300 K-40

N5 G200

N6 T2 M6 (predrilling start point, drill R10)

N7 G81 Y2 Z-20 F200 S3000 M3

N8 G22 N=9992

N9 T3 M6 (clearing out the pocket, mill R8)

N10 S2500 M3

N11 G201 Y0.1 Z-20 B2 I50 K8 F200 F2=100 N1=9991 N2=9992

N12 G203 X40 Y40 Z0 N1=9993

N13 G208 X220 Y220 I30 (pocket contour)

N14 G204

N15 G205 X100 Y80 N1=9994

N16 G208 X-30 Y30 J-1 (Island 1)

N17 G206

N18 G205 X190 Y80 N1=9995

N19 G91

N20 Y50 (Island 2)

N21 X40 Y-50

N22 G90

N23 G206

N24 G205 X150 Y130 N1=9996

N25 G2 I150 J150 (Island 3)

N26 G206

N27 G205 X110 Y210 N1=9997

N28 G208 X-40 Y40 J-1 B1=135 (Island 4)

N29 G206

G-FUNCTIONS

N30 G205 X180 Y200 N1=9998
N31 G91
N32 Y30
N33 X20 (Island 5)
N34 X30 Y-30
N35 G90
N36 G206
N37 G202
N38 T4 M6 (clearing out the pocket, mill R8)
N39 F200 S2200 M3 |
N40 G22 N=9993 |
N41 G22 N=9994 |
N42 G22 N=9995 | (finishing)
N43 G22 N=9996 |
N44 G22 N=9997 |
N45 G22 N=9998 |
N46 M30

6. Specific G-Functions for macros

6.1 Overview:

Error message functions

- G300 Programming error messages
- G301 Error in a program or macro

Executable functions

- G302 Overwriting radius compensation parameters.
- G303 M19 with programmable direction

Query functions

- G319 Query actual technology data
- G320 Query actual G-data
- G321 Query tool data
- G322 Query machine constant memory
- G324 Query G-group
- G325 Query M-group
- G326 Query actual position
- G327 Query operation mode
- G329 Query offset from kinematics model

Write functions

- G331 Write tool data
- G339 Write offset in kinematics model

Formatted write functions

- G350 Display window
- G351 Write to file

6.2 Error message functions

6.2.1 G300 Programming error messages

Purpose

Setting error messages during the execution of universal programs or macros.

Format

N... G300 [{D...}]{D1=...}

Parameter

```
G   Program error call  
D   P Error message number  
D1= R Error message number
```

Notes and usage

D are general milling error messages (P), D1= are error messages (R) in turning mode (G36)

An error message may be given during program execution.

The error messages only cover the existing P and R-errors (refer to Machine Manual).

Examples

Example: Setting an error message if a programmed angle is not allowed.

```

N9999 (Macro for calculation of table rotations)
N11 (input parameter: E4: phi)
N100
N110 G29 I1 E30 N=180 E30=(E4>360)
N120 G29 I1 E30 N=210 E30=(E4<0)
N150 G29 I1 E30 N=290 E30=1
N160
N170 (error message: phi>360)
N180 G300 D190 (programmed value > maximum value)
N190
N200 (error message: phi<0)
N210 G300 D191 (programmed value < minimum value)
N220
N290

```

Explanation:

```

N11:   E4 is the input value for angle phi
N110:  Compare if E4 > 360 degrees. If so, jump to N180
N120:  Compare if E4 < 0 degree. If so, jump to N210
N150:  Jump to 290 (0 <= E4 <= 360 degrees)
N180:  Error message: programmed value > maximum value
       Program should be ended and a modified E4 be entered
N210:  Error message: programmed value < minimum value
       Program should be ended and a modified E4 be entered
N290:  End program

```

6.2.2 G301 Error in program or macro that just has been read in.**Purpose**

Error in program or macro block that just has been read in.

Format

N... G301 (O... Wrong original block)

Parameter

none

Notes and usage

When the controller retrieves a program block or macro block and discovers an error it activates G301
Function G301 can only be active in an error stopped program or macro.

This function cannot insert in MDI.

The error texts are O errors. (Refer to Machine Manual).

Example

The program is stored on hard disk.
Program is made with a MC84=0.

```
N9999 (Program)
N1 G17
N2 G57
N3 T1 M6
N4 F200 S1000 M3
..
N99 M30
```

Error stops program in RAM.
Zero point shift extension MC84 > 0 is active.

```
N9999 (ERR*)(Program ...)
N1 G17
N2 G301 (O138 G57)
N3 T1 M6
N4 F200 S1000 M3
..
N99 M30
```

Explanation:

N2: G301 explains that the program is false. G57 must be G54 I3

The false program can be activated. When passing the block G301 the controller stops and gives the following error text P33 (Modify block converted to connect). The block containing G301 must be changed before restarting.

6.3 Executable functions

6.3.1 G302 Overwriting radius compensation parameters

Purpose

The G302 function overwrites the active tool parameters during execution. The tool parameters in the tool memory are not changed.

In this version, only the O parameter for tool orientation can be overwritten.

For description refer to paragraph 12.8 in chapter "Turning mode" on page 560.

6.3.2 G303 M19 with programmable direction

Purpose

M19 with programmable direction (CW or CCW).

Format

N... G303 M19 D... I2=...

Parameter

G	M19 with programmable direction
D	Angle oriented spindle stop
M	Machine function
I2=	Direction 3=CW 4=CCW

Notes and usage

Only M19 can be programmed.

Default for I2=3

Example

Example: Stop spindle with M19.

N100 G303 M19 D75 I2=4

Explanation:

N100: Orientates spindle stop
 Angle 75 degrees
 CCW

6.4 Query functions

6.4.1 G319 Query actual technology data

Purpose

Query active F (Feed), S (Speed), S1 (Cutting speed/rotational speed) or T (Tool number).

Format

N... G319 I1=.. E... {I2=..}

Parameter

```
G   Read actual technology data
E   E-parameter
I1= 1-8 (F,S,T,S1,F1,F3,F4,F5)
I2= 0=programmed, 1=actual
```

Possible function:

I1=1	Feed (F)
I1=2	Speed (S)
I1=3	Tool number (T)
I1=4	Cutting speed/speed (S1=) (only turning)
I1=5	Constant cutting feed (F1= by G41/G42)
I1=6	In depth feed (Infeed F3=)
I1=7	In plane feed (F4=)
I1=8	Feed for rotary axes (F5=)
I2=0	Programmed value (default)
I2=1	Actual value.

Associated functions

Non-modal

Notes and usage

READING OF ADRESSES WITHOUT DATA

If the address not exist, the E-parameter contains the number -999999999.

Example

Example: query active feed and storing the data in E-parameter 10.

N... G319 I1=1 E10 I2=0

Explanation:

I1=1 query feed.

E10 then contains the value

6.4.2 G320 Query current G data

Purpose

Query address value of current modal G function and save this value in the E parameter provided for this purpose.

Format

N... G320 I1=.. E...

Parameter

G	Read G-group
E	E-parameter
I1=	G-group (1,2,etc.)

Associated functions

Non-modal

Notes and usage

DEFAULTS

All values are initialised when the machine is started. Most parameters are set on zero.

READING ACTIVE MODAL G-FUNCTIONS

G324 can be used to query whether a G function is active.

Particular information can always be queried with G320.

RESULT DIMENSION

The unit of the result is mm or inches. Degrees for angles.

SELECTION NUMBER

G-function		result	default
I1=selection number		min—max.	
G7	Tilting working plane		
	1 Angle of rotation A-axis	-180--180°	0
	2 Angle of rotation B-axis	-180--180°	0
	3 Angle of rotation C-axis	-180--180°	0
G8	Tilting tool orientation		
	4 Angle of rotation A-axis	-180--180°	0
	5 Angle of rotation B-axis	-180--180°	0
	6 Angle of rotation C-axis	-180--180°	0

SPECIFIC G-FUNCTIONS FOR MACROS

G9	Defining pole position point		
7	Pole coordinate X-axis		0
8	Pole coordinate Y-axis		0
9	Pole coordinate Z-axis		0
Result from G17, G18, G19, G180 and G182			
10	First main axis	1--3	
11	Second main axis	1--6	
12	Tool axis	1--3	
		1=X, 2=Y, 3=Z, 4=A, 5=B, 6=C	
G25	Feed- and speed override active		
13	Feed- and speed override active	0	
G26	Feed- and speed override not active		
13	Feed- and speed override not active	1--3	
		1=F=100%, 2=S=100%, F und S=100%	
G27	Positioning functions		
14	Feed movement (I3=)	0	
15	Rapid movement (I4=)	0	
16	Positioning logic (I5=0)	0	
17	Acceleration reduction (I6=)	100%	
18	Contour tolerance (I7=0)	MC765	
G28	Positioning functions		
14	Feed movement (I3=)	0--1	
15	Rapid movement (I4=)	0--1	
16	Positioning logic (I5=0)	0--1	
17	Acceleration reduction (I6=)	5—100%	
18	Contour tolerance (I7=0)	0—10.000µm or MC765	
G39	Activate tool offset		
19	Tool length offset (L)		0
20	Tool radius offset (R)		0
G52	Palettes zero point shift		
21	Zero point shift in X-axis		0
22	Zero point shift in Y-axis -		0
23	Zero point shift in Z-axis -		0
24	Zero point shift in A-axis -		0
25	Zero point shift in B-axis -		0
26	Zero point shift in C-axis -		0
G54	Standard zero point shift		
27	Zero point shift in X-axis -		0
28	Zero point shift in Y-axis -		0
29	Zero point shift in Z-axis -		0
30	Zero point shift in A-axis -		0
31	Zero point shift in B-axis -		0
32	Zero point shift in C-axis -		0
33	Angle of rotation	-180--180°	0

G92/G93	incremental or absolute zero point shift	
34	Zero point shift in X-axis -	0
35	Zero point shift in Y-axis -	0
36	Zero point shift in Z-axis -	0
37	Zero point shift in A-axis -	0
38	Zero point shift in B-axis -	0
39	Zero point shift in C-axis -	0
40	Angle of rotation	-180--180° 0
United zero point shift (G52 + G54 + G92/G93)		
41	Zero point shift in X-axis -	0
42	Zero point shift in Y-axis	0
43	Zero point shift in Z-axis -	0
44	Zero point shift in A-axis	0
45	Zero point shift in B-axis -	0
46	Zero point shift in C-axis -	0
47	Angle of rotation	-180--180° 0
G72	Mirror image and scaling not active	
48	Scaling (factor or %) plane (A4=)	1
49	Scaling (factor or %) tool axis (A4=)	1
50	Mirror image in X-axis	1
51	Mirror image in Y-axis	1
52	Mirror image in Z-axis	1
53	Mirror image in A-axis	1
54	Mirror image in B-axis	1
55	Mirror image in C-axis	1
G73	Mirror image and scaling active	
48	Scaling (factor or %) plane (A4=)	0—100% or factor
49	Scaling (factor or %) tool axis (A4=)	0—100% or factor
		Depended of MC714
	MC714 0= Machining plane (factor)	
	1= Machining plane (percent eel)	
	2= all linear axes (factor)	
	3= all linear axes (percent)	
50	Mirror image in X-axis	-1--1
51	Mirror image in Y-axis	-1--1
52	Mirror image in Z-axis	-1--1
53	Mirror image in A-axis	-1--1
54	Mirror image in B-axis	-1--1
55	Mirror image in C-axis	-1--1
System axes number determinate by machine constants (MC103, MC105, etc.).		
56	First main axis	0--6
		0=not active, 1=X, 2=Y, 3=Z, 4=A, 5=B, 6=C
57	Second main axis	0--6
58	Tool axis	0--6
59	First rotation axis	0--6
60	Second rotation axis	0--6
61	Third rotation axis	0--6

Example

Example: Query of Address of G-function (I1=) and store of the value in E-parameter 10.

```
N11 G320 I1=10 E11
N12 G320 I1=11 E12
N12 G320 I1=12 E13
```

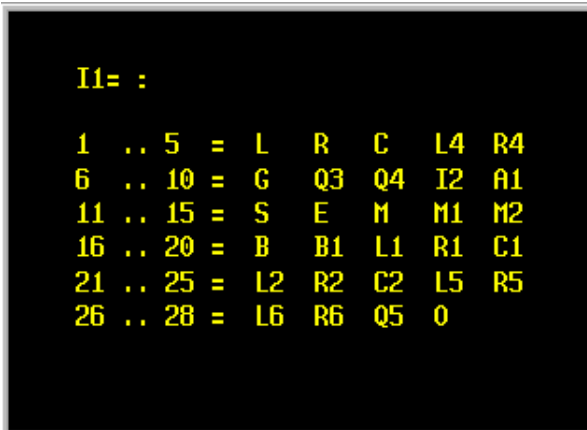
Explanation:

```
N11:  I1=10  Query first main axis
      E11 contains the result
      E11 =1      X-axis is first main axis.
N12:  I1=11  Query second main axis
      E12 contains the result
      E12 =2      Y-axis is second main axis.
N13:  I1=12  Query tool axis
      E13 contains the result
      E13 =1      Z-axis is tool axis.
```

6.4.3 G321 Query tool data

Purpose

Query tool table.



I1= :					
1 .. 5	=	L	R	C	L4 R4
6 .. 10	=	G	Q3	Q4	I2 A1
11 .. 15	=	S	E	M	M1 M2
16 .. 20	=	B	B1	L1	R1 C1
21 .. 25	=	L2	R2	C2	L5 R5
26 .. 28	=	L6	R6	Q5	0

Format

N... G321 T.. I1=.. E...

Parameter

```
G   Read tool data
T   Tool number
E   E-parameter
I1= Tool address (1=L .. 29=0)
```

Associated functions

Non modal

Notes and usage

TOOL NUMBER AND POSITION

The Tool number (T) must be known. The position (P) in the tool table cannot be queried.

E-PARAMETER

One E-Parameter is taken with the value from the tool table.

READING OF THE TOOL TABLE VALUES WITHOUT DATA

If The E-Parameter contains the number -999999999, the address in the tool table is empty.

CLASSIFICATION

I1=1	L	Length
I1=2	R	Radius
I1=3	C	Corner radius
I1=4	L4=	Length oversize
I1=5	R4=	Radius oversize
I1=6	G	Graphics
I1=7	Q3=	Type
I1=8	Q4=	Number of cutting edges
I1=9	I2=	Cutting direction
I1=10	A1=	Approach angle
I1=11	S	Size
I1=12	E	Status
I1=13	M	Initial tool life
I1=14	M1=	Actual tool life
I1=15	M2=	Tool life monitoring
I1=16	B	Breakage tolerance
I1=17	B1=	Breakage monitoring
I1=18	L1=	First extra length
I1=19	R1=	First extra radius
I1=20	C1=	First extra corner radius
I1=21	L2=	Second extra length
I1=22	R2=	Second extra radius
I1=23	C2=	Second extra corner radius
I1=24	L5=	Wear tolerance length
I1=25	R5=	Wear tolerance radius
I1=26	L6=	Offset length
I1=27	R6=	Offset radius
I1=28	Q5=	Breakage monitoring cycle (0-9999)
I1=29	O	Tool orientation (only turning)

Example

Example: Program queries the tool table.

```

N30 G321 T10 I1=1 E1
N40 G321 T10 I1=2 E10
N50 G321 T10 I1=3 E20
N60 G321 T10 I1=4 E2
N70 G321 T10 I1=5 E11
N80 E3=E1+E2
N90 E12=E10+E11

```

Explanation:

N30: G321 Read command
 T (tool number)
 I1= Information about the tool address
 E1 is E-parameter
 L (tool length) is set in E-parameter 1
 N40: R (tool radius) is set in E-parameter 10
 N50: C (corner radius) is set in E-parameter 20 (If C has no value, E20=-999999999 is set)
 N60: L4 (length oversize) is set in E-parameter 2
 N70: R4 (radius oversize) is set in E-parameter 11
 N80: The correct tool length (E3) is L+L4 (E1+E2)
 N90: The correct tool radius (E12) is R+R4 (E10+E11)

6.4.4 G322 Query machine constant memory

Purpose

To read out a machine constant value and store it in the appropriate E-parameters.

Format

N... G322 E.. N1=...

Parameter

G	Read machine constant memory
E	E-parameter
N1=	Machine constant number

Associated functions

Non modal

Notes and usage

READING OUT A MACHINE CONSTANT WITHOUT VALUE

When invisible addresses are read from the machine constant table, the E-parameter remains unchanged.

Examples

Example: Read out machine constant 103 and store value in E-parameter 10.

N... G322 N1=103 E10

E10 contains the number of machine constant 103

Example: Universal program blocks, which can be used for both zero point, table types.

```

N50 G322 N1=84 E10
N60 G29 E1 N=90 E1=E10>0
N70 G150 N1=57 X7=E1 Z7=E6
N80 G29 E1 N=100 E1=1
N90 G150 N1=54.3 X7=E1 Z7=E6
N100 ..

```

Explanation:

```

N50: Machine constant 84 is set in E10
N60: Compare if MC84 > 0. If so, jump to N90
N70: Store the zero point shift table ZO.ZO
N80: Jump to N100
N90: Store the zero point shift table ZE.ZE

```

6.4.5 G324 Query G-group

Purpose

Query current **modal** G-function and stores with this value in the E-Parameters preprogrammed for this purpose.

Format

N... G324 I1=.. E...

Parameter

G	Read G-group
E	E-parameter
I1=	G-group (1,2,etc.)

Associated functions

Non modal

Notes and usage

READ OUT OF GROUP WITHOUT DATA

If the group or the G-function not exists, the E-parameter is unchanged.

Group classification

Group	
I1=	G-function
1	G0, G1, G2, G3, G6, G9
2	G17, G18, G19
3	G40, G41, G42, G43, G44, G141
4	G53, G54, G54_I, G55, G56, G57, G58, G59
5	G64, G63
6	off, G81, G83, G84, G85, G86, G87, G88, G89, G98
7	G70, G71
8	G90, G91
10	G94, G95
11	G96, G97 (only turning)
12	G36, G37 (only turning)
13	G72, G73
14	G66, G67
15	off, G39
16	G51, G52
17	G196, G199
19	G27, G28
20	G25, G26, G26_S, G26_F_S

21	off, G9
22	G202, G201
24	G180, G182, G180_XZC
27	off, G7
28	off, G8
29	G106, G108 (From version V501).

Result

In general is the result equal to the value of the modal G-function.

For example: G324 I1=3 gives, when G40 is active, as result the value 40.

Exceptions are:

Off gives value 0.
G26_S, G26_F_S gives 26.
G54_I gives 54.nn, where nn is the index.
G180_XYZ gives 180.

Example

Example: selection of the G-function (I1=2) and storage of the value in E-parameter 10.

N... G324 I1=2 E10

Explanation:

I2=2: query group 2 G-function
E10 holds the result
E10 =17 G17 is active.
E10 =18 G18 is active.
E10 =19 G19 is active.

6.4.6 G325 Query M group

Purpose

Query current modal M-function and store this value in the E-Parameter pre-programmed for this purpose.

Format

N... G325 I1=.. E...

Parameter

G	Read M-group
E	E-parameter
I1=	M-group (1,2,etc.)

Associated functions

Non modal

Notes and usage

READ OUT OF GROUP WITHOUT DATA

If the group or the M-function does not exist, the E-parameter is unchanged.

Group classification

Group	M-function
I1=	
1	off, M5, M3, M4, M19
2	off, M40, M41, M42, M43, M44
3	M9, M7, M8
4	off, M17, M18, M19
5	off, M10, M11
6	off, M22, M23
7	off, M32, M33
8	off, M55
9	off, M51, M52
10	off, M53, M54
11	off, M56, M57, M58
12	off, M72, M73
13	off, M1=..

MEANING M-FUNCTIONS

Some of these M-functions are basis M-functions and are described in the M-functions chapter. The other are machine dependent M-functions. Please refer to the machine builder handbook for a description.

COMBINED M_FUNCTIONS (M13 and M14)

M13 and M14 are combined M-functions. ($M13 = M3 + M8$). These functions are determinate by two blocks.

N... G325 I1=1 E10.

N... G325 I1=3 E11

When $E10=3$ and $E11=8$, then M13 is active.

Result

In general is the result equal to the value of the modal M-function.

For example: G324 I1=2 gives, when M40 is active, as result the value 40.

Exceptions are: Off gives value 0.

Example

Example: selection of the M-function (I1=1) and storage of its value in E-parameter 10.

N... G325 I1=1 E10

Explanation:

I2=1: query group 1 M-function

E10 holds the result

E10 =5 M5 is active.

E10 =3 M3 is active.

6.4.7 G326 Query actual position**Purpose**

To read out the actual axes-positions values and store it in the appropriate E-parameters.

Format

N... G326 {X7=..} {Y7=..} {Z7=..} {A7=..} {B7=..} {C7=..} {I1=..} {I2=..}

Parameter

```
G    Read actual position
X7=  E-parameter for X-position
Y7=  E-parameter for Y-position
Z7=  E-parameter for Z-position
A7=  E-parameter for A-position
B7=  E-parameter for B-position
C7=  E-parameter for C-position
I1=  0=Workp. 1=Machine 2=RPF
I2=  0=programmed, 1=actual
```

I1=	0	Position to work piece zero point (Default)
	1	Position to machine zero point
	2	Position to reference point
	3	Total zero point shift (without IPLC shift).
I2=	0	Programmed value (default)
	1	Current value

Notes and usage**READING OUT OF NOT EXISTENT AXES**

When an axis not exist the contents of the E-parameter is filled with -999999999.

READING OUT BY GRAPHICAL SIMULATION

By graphical simulation only the X, Y and Z can be read out. The E-parameters for the rotating axes stays zero.

Examples**Example 1:**

Read out actual axes-position von X, Y and Z and store the values in E-parameters 20, 21 and 22.

```
N... G326 X7=20 Y7=21 Z7=22
```

E20 contains the actual X-axis-position.

Example 2:

Program continuation after a universal pocket cycle.

```
N30 G202  
N40 G326 X7=20 Y7=21  
N50 G29 E1 N=90 E1=E20>100  
N60 G29 E1 N=90 E1=E20<-100  
N70 G0 X-110  
N80 G0 Y 100  
N90 ..
```

Explanation:

N30: End pocket cycle
N40: Unknown actual End-position von X and Y
N50: Actual X-position >100, then jump to N90
N60: Actual X-position <-100, then jump to N90
N70: G0 movement to X-110, if the actual X-position is situated between 100 and -100.
On this manner for example an obstacle can be rounded.
N80: Further turn aside movement
N90:

6.4.8 G327 Query operation mode

Purpose

To scan the current operating mode and store this value in the E parameter provided.

Format

N... G327 I1=.. E...

Parameter

```
G   Read operation mode
E   E-parameter
I1= Active mode (1-6)
```

Associated functions

Non modal

Arrangement of group

Group		
I1=	Operating mode	
1	Easy Operate	0 = not active, 1=active
2	Single record	0 = not active, 1=active
3	Graphic	0 = not active, 1=active
4	Test run	0 = not active, 1=active
5	Search	0 = not active, 1=active
6	Demo	0 = not active, 1=active

Examples

Example: Fetch operating mode (I1=1) and store the value in E parameter 10.

N... G327 I1=1 E10

Explanation:

I1=1: Check whether Easy Operate is active.
E10 contains the result: 0= not active, 1=active.

6.4.9 G329 Query programmable kinematic model

Purpose

Query programmable kinematic elements and save these values in the E parameter provided for this purpose.

Format

N... G329 N1=.. E...

Parameter

G	Read offset from kinematic model
E	E-parameter
N1=	Progr.kinematic element nr.(1-10)

Notes and usage

Refer to chapter: 14.1.3 "kinematic model" on page.682.

PROGRAMMABLE KINEMATIC ELEMENTS

A kinematic element is defined by a group of 4 machine constants.

The machine builder can enter whether a particular kinematic element is programmable. To do this, the third machine constant of the group (MC602, MC606, etc.) must have the value 2.

The values of these programmable kinematic elements can be programmed using G339. In doing this, the value of the fourth machine constant of the group (MC603, MC607, etc.) is overwritten.

PROGRAMMABLE KINEMATIC ELEMENT NUMBER

Defines the number of the programmable element of the kinematic model to be queried. The value lies between 1 and 10.

Example

Read a programmable kinematic element and save the value in the E parameter.

N... G329 N1=1 E10

Explanation:

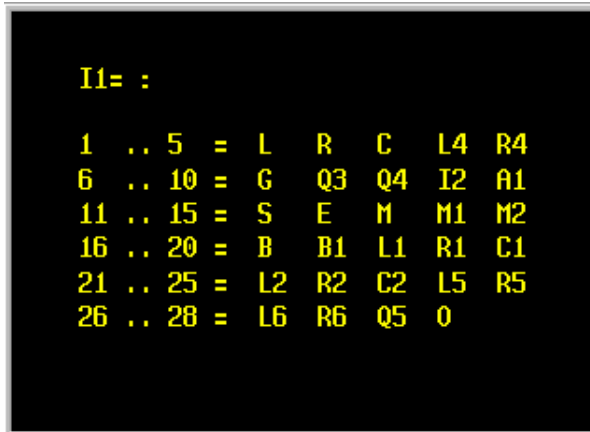
E10 contains the contents of programmable kinematic element 1 (in mm or inches).

6.5 Write functions

6.5.1 G331 Write tool data

Purpose

Write from values in the tool table.



The screenshot shows a CNC control interface with a black background and yellow text. At the top, it displays 'I1= :'. Below this, there is a table of tool data with 6 rows and 5 columns. The first column contains tool numbers (1, 6, 11, 16, 21, 26), the second column contains ranges (.. 5, .. 10, .. 15, .. 20, .. 25, .. 28), the third column contains letters (L, G, S, B, L2, L6), the fourth column contains letters (R, Q3, E, B1, R2, R6), the fifth column contains letters (C, Q4, M, L1, C2, Q5), the sixth column contains letters (L4, I2, M1, R1, L5, 0), and the seventh column contains letters (R4, A1, M2, C1, R5, 0).

Tool Number	Range	Letter 1	Letter 2	Letter 3	Letter 4	Letter 5
1	.. 5	L	R	C	L4	R4
6	.. 10	G	Q3	Q4	I2	A1
11	.. 15	S	E	M	M1	M2
16	.. 20	B	B1	L1	R1	C1
21	.. 25	L2	R2	C2	L5	R5
26	.. 28	L6	R6	Q5	0	

Format

N... G331 T.. I1=.. E...

Parameter

G Write tool data
T Tool number
E E-parameter
I1= Tool address (1=L .. 29=0)

Associated functions

Non modal

Notes and usage

TOOL NUMBER AND POSITION

The tool number (T) must be known. The position (P) in the tool table cannot be changed.

E-PARAMETER

One E-parameter is written in the tool table.

WRITING IN THE TOOL TABLE WITHOUT DATA

If the E-parameter contains the value -999999999, the address in the tool table becomes empty.

NEW INFORMATION ACTIVATING

The changed tool information must be activated again following the writing. (T.. M67)

CLASSIFICATION

I1=1	L	Length
I1=2	R	Radius
I1=3	C	Corner radius
I1=4	L4=	Length oversize
I1=5	R4=	Radius oversize
I1=6	G	Graphics
I1=7	Q3=	Type
I1=8	Q4=	Number of cutting edges
I1=9	I2=	Cutting direction
I1=10	A1=	Approach angle
I1=11	S	Size
I1=12	E	Status
I1=13	M	Initial tool life
I1=14	M1=	Actual tool life
I1=15	M2=	Tool life monitoring
I1=16	B	Breakage tolerance
I1=17	B1=	Breakage monitoring
I1=18	L1=	First extra length
I1=19	R1=	First extra radius
I1=20	C1=	First extra corner radius
I1=21	L2=	Second extra length
I1=22	R2=	Second extra radius
I1=23	C2=	Second extra corner radius
I1=24	L5=	Wear tolerance length
I1=25	R5=	Wear tolerance radius
I1=26	L6=	Offset length
I1=27	R6=	Offset radius
I1=28	Q5=	Breakage monitoring cycle (0-9999)
I1=29	O	Tool orientation (only turning)

The tool commentary cannot be changed.

Example

```

N10 E5=100 (Tool length)
N11 E6=10 (Tool radius)
N12 E7=-999999999 (Tool corner radius)
N13 E8=0 (Length oversize)
N14 E9=0 (Radius oversize)

```

```

N..
N20 G331 T10 I1=1 E5
N21 G331 T10 I1=2 E6
N22 G331 T10 I1=3 E7
N23 G331 T10 I1=4 E8
N24 G331 T10 I1=5 E9

```

```

N30 T10 M67
N..
N40 E8=0.3 (Length oversize)
N41 G331 T10 I1=4 E8

```

```

N50 T10 M67

```

Explanation:

```

N10:  L (tool length) is set in E-parameter 5
N11:  R (tool radius) is set in E-parameter 6
N12:  C (tool corner radius) will be stored in E-parameter 7 (If C has no value, E7= must
      be set to -999999999)
N13:  L4 (length offset) is set in E-parameter 8
N14:  R4 (radius offset) is set in E-Parameter 9

N20:  L (tool length) writing of E-parameter 5 in the tool table
N21:  R (tool radius) writing of E-parameter 6 in the tool table
N22:  C (tool corner radius) writing of E-parameter 7 in the tool table
N23:  L4 (length offset) writing of E-parameter 8 in the tool table
N24:  R4 (radius offset) writing of E-parameter 9 in the tool table

N30:  The tool must be activated once more with the changed information.

N40:  L4 (length offset) E-parameter 8 is set to 0.3
N41:  L4 (length offset) writing of E-parameter 8 in the tool table

N50:  Tool must be activated once more with the changed information.

```

6.5.2 G339 Write programmable kinematic model

Purpose

Write programmable kinematic elements.

Format

N... G339 N1=.. E... {I1=...}

Parameter

```
G   Write offset in kinematic model
E   E-parameter
N1= Progr.kinematic element nr.(1-10)
I1= Write mode      (0=inc, 1=abs)
```

Notes and usage

Refer to paragraph: 14.1.3 "kinematic model" on page 682.

PROGRAMMABLE KINEMATIC ELEMENTS

A kinematic element is defined by a group of 4 machine constants.

The machine builder can enter whether a particular kinematic element is programmable. To do this, the third machine constant of the group (MC602, MC606, etc.) must have the value 2.

The values of these programmable kinematic elements can be programmed using G339. In doing this, the value of the fourth machine constant of the group (MC603, MC607, etc.) is overwritten.

PROGRAMMABLE KINEMATIC ELEMENT NUMBER

Defines the number of the programmable element of the kinematic model to be written. The value lies between 1 and 10.

NOTATION

"Incremental" notation (basic setting) means that the programmed value is added to the existing value.

"Absolute" notation means that the existing value is overwritten by the programmed value.

The programmed values remain in the kinematic model and are not reset following M30, <Cancel program> or <Reset CNC>.

Example: Write a programmable kinematic element.

A circular table is defined in the kinematic model. This circular table is defined by two kinematic elements for each X axis. The first is defined by the machine builder and determines the position of the circular table. The second is a programmable element. Using this, the exact position can be corrected in the kinematic model after measurement.

```

N100 G145... (measure)
N105 (calculate all parameters)
N110 G339 N1=1 E10 I1=1
  
```

Explanation:

```

N100   Measuring the exact position.
N110   The contents of E10 are written to the first programmable kinematic element.
  
```

6.6 Formatted write functions

Previously, only write functions to the internal memory were possible.

Using the formatted write function, it is now possible:

- to write to the screen
- to write to the file on the hard disc
- to fill an array
- to read a number from an array.

Configuration file

Configuration files are required for these actions to describe how and where to write or read. These configuration files are saved on the hard disk and are read when the system is started.

Two configuration files are possible.

- 1) File to define an array and fill it with basic settings.
ARRnnnnn.CFG
nnnnn is the file number from 1 to 99999.
- 2) File to define a print file.
D:\STARTUP\CYCLES\FORMnnnnn.CFG.
nnnn is the file number from 1 to 9999.

Overview:

- 6.1 File to define an array and fill it with basic settings (Array).
- 6.2 E-parameter array
- 6.3 Configuration file to define a file or window (display/input).
- 6.4 Writing to a window G350
- 6.5 Writing to a file G351

6.6.1 File to define an array and fill it with basic settings.

An array is defined with a configuration file. This is activated when the system is started.

A maximum of 10 arrays can be defined. End users can define files themselves.

The maximum array size is 5000 elements.

An element can be read from the array with arrayread (nnnnn, row, column).

If an element is read that does not exist, the value 999999999 is returned.

Description of array configuration file:

```

;Comments start with ';'
;
;Sections:
;[element]
;row    =    row number    defines an element in the array
;                        where row number = [1|...|9999]
;col    =    column number  where column number = [1|...|9999]
;                        row * column <= 5000
;val    =    value          where value = real number (double)
;

```

Example: Array configuration file:

ARRnnnnn.CFG

```

[element]
row    =    1
col    =    1
val    =    0          ; element (1,1).=.0

[element]
row    =    3
col    =    66
val    =    397.01     ; element (3,66) = 397.01

[element]
row    =    9999       ;maximum row size
col    =    9999
val    =    -123456789.123456789

```

6.6.2 E-Parameter Array

Several arrays can be filled using a configuration file. These arrays can be read with E parameters during execution. For unbalance detection, the calibration curves are read and interpolated in this manner.

arrayread (arraynumber, row, column)

Where:

arraynumber	is the number of the array. Every array has its own configuration file. Arraynumber between 1 and 89999.
Row	is the row number in the array that is to be read. Row between 1 and 999999.
Column	is the position in the row of the array that is to be read. Column between 1 and 999999.

Fixed arrays can be read with the arrayread function. The arrays are filled from a configuration file D:\STARTUP\CYCLES\ARRnnnnn.CFG).

Empty 'elements' in the array have the value <-999999999>.

Example

```
arrayread
E300 = arrayread(100,1,2)
E300 has the value of array 100, row 1, column 2.
```

6.6.3 Configuration file to define a file or window (display/input).

A print file is defined with a configuration file. This is activated when the system is started.

A maximum of 10 files can be defined. End users can define files themselves.

The file size is unlimited.

Description of print file configuration file:

```

;Commentary starts with a ';'
;
;Sections:
Only for one window:
;[window]                defines present window
;number=      windowId   where windowId =      1 = output, centre, 5x40
;                2 = input, centre, 1x40;
;                3 = graphic, above dashboard
;
;[file]                defines file (only for G351')
;name =      file name  where filename is 8.3 ASCII characters
;                The directory is always D:\STARTUP\
;
;[string]                defines position and content of the block
;line =      line number where line number = [1|...|n] basic setting = 1
;position=    position number where position number = [1|...|n] basic setting = 1
;gb =      "string"      where block is <n> ASCII characters
;d =      "string"
;
;                Texts are defined for various languages
;                Code gb=, d=, f= .. or defined independent of language with: txt =
;
;[value]                defines position, format and E parameter of the value
;line =      line number
;position=    position number
;eparam=      E parameter where E parameter number = [1|...|1250]
;form =      digit decimal where digit decimal = <digits>.<decimals>
;sign =      yesNo       where yesNo = y = space for sign
;                n = no space for sign
;
Only for input window:
;[input]                defines position, format and E parameter before an input field
;                only for G350 and windowId = 2
;                only one [input] section is allowed
;
;line =      line number
;position=    position number
;eparam=      E parameter number where E parameter number = [1|...|1250]
;form =      digitDecimal where digitDecimal = <digits>.<decimals>
;sign =      yesNo       where yesNo = y =space for sign
;                n = no space for sign
;

```


6.6.4 G350 Writing to a window

Purpose

Specific lines and values can be written to a window using E parameters and a configuration file. In addition, a particular input can be expected. For unbalance detection, the result can be displayed to the operator in this way.

Format

N... G350 N1=.. {I1...}

Parameter

```
G    Write to window
N1=  Configuration file number
I1=  Window (0=closed, 1=open)
```

N1= Defines the configuration file 'D:\STARTUP\CYCLES\IFORMnnnn.CFG' that is used for the format, lines and E parameters that are written. File number between 1 and 8999.

I1= 0 = window not visible. Setting on switch-on:
1 = window visible.

Notes and application

G350 can be used to make a previously defined window visible. The texts in the window are fixed, the values are continuously updated according to the defined E parameters. When an input field is defined, the program waits until the entry is made and <Start> is pressed. Only one entry window can be active at any one time.

Up to now, 3 windows are defined:

Number	Window type	Mode	Position	Size
1	Display	Manual Automatic	Right side of screen Top 'Dashboard'	14 lines, 35 characters
2	Input	Manual Automatic	Right side of screen Top 'Dashboard'	1 line, 35 characters
3	Graphics	Manual Automatic	Up to machine function keys Top 'Dashboard'	14 lines, 70 characters

Please see also the configuration file.

The window also appears in graphics, but not during block search.
the window becomes invisible following M30 and <Cancel program>.

Example Writing to a window

N1 E300=8 Hole number
 N2 E301=5 Number
 N3 E350=6 Suggested input value

 N10.. G350 N1=6789 Write to window
 File D:\STARTUP\CYCLES\FORM6789.CFG is used
 N

; The window is:

```

+-----+
|Drilling pattern|
+-----+
|Holes  8  Number  5|
+-----+
|Restart at number   |||
+-----+
  
```

E350 is changed following user input.

Display window configuration file

FORM6789.CFG

[window]
 number = 1 ;Uses window number 1 of the available windows.

[string]
 line = 1
 gb = "drilling pattern"
 d = "Bohrbild"

[string]
 line = 2
 gb = '-----'
 d = '-----'

[string]
 line = 3
 gb = "holes"
 d = "Locher"

[value]
 line = 3
 position = 8 ;Print value in field at position 8 and onwards
 eparam = 300 ;E parameter E300 is given the value
 form = 3.0 ;format 3 figures and 0 decimals
 sign = n ;No space reserved for sign

[string]
 line = 3
 position = 17 ;Print value at position 17 and onwards
 gb = "number"
 d = "Nummer"

[value]
 line = 3
 position = 26 ;Print value in field at position 26 and onwards
 eparam = 301 ;E parameter E301 is given the value

```

form    =      3.0    ;format 3 figures and 0 decimals
sign    =      n      ;No space reserved for sign

[string]
line    =      4
gb      =      '-----'
d       =      '-----'

[string]
line    =      5
gb      =      "restart at number:"
d       =      "Neustart bei Nummer:"

[input]
line    =      5
position =      26    ;Define input field at position 26 and onwards
eparam  =      350    ;E parameter E350 suggests an input value
                        ;and later received the operator input
form    =      3.0    ;format 3 figures and 0 decimals
sign    =      n      ;No space reserved for sign

```

6.6.5 G351 Writing to a file

Purpose

Specific lines and values can be written to a text file in D:\Startup\ using E parameters and a configuration file. This can be used to create the calibration curves for unbalance detection.

Format

N... G351 N1=.. {N2=...} {I1=...}

Parameter

```
G    Write to file
N1=  Configuration file number
I1=  0=Add, 1=Overwrite
```

- N1= Defines the configuration file <'Directory'\FORMnnnn.CFG> that is used for the format, lines and E parameters that are written. File number between 1 and 9999. The directory can be any 'Cycle Design' directory. The configuration file is the same as for writing to a window, but 'section' [window] and [input] are ignored.
- I1= States whether the data is to be inserted at the end of an existing file or whether a file that may exist is to be overwritten. Basic setting <0> for insertion.

Notes and application

G351 writes the lines and values of the configuration file and E parameters to the hard disk.

A maximum of 50 lines of 255 characters can be written at the same time.

The file is not written during graphics or block search.

Example Listing measurement data and writing to a file.

The radius of a pocket is measured in the program

The following data available in the E parameters must be listed:

N10 (measurement programmed in blocks N12 to N16)
 N11 (in this case as example of just the results from e.g. measurement cycle G145)
 N12 E50=34.1 (setpoint) (entered)
 N13 E51=34.05 (lower tolerance) (entered)
 N14 E52=34.15 (upper tolerance) (entered)
 N15 E53=34.108 (actual value) (measured)
 N16 E54=0.008 (difference) (calculated)

N20 G351 N1=0002 I1=0 (write file)
 File D:\STARTUP\CYCLES\FORM0002.CFG is used.
 I1=0 is insert

The file messdat.txt is:

```
Radius
Setpoint =      34.1
Lower tolerance =34.5
Upper tolerance =34.5
Actual value =   34.108
Difference =      0.008
*****
```

Configuration file for listing measurement data

FORM0002.CFG

```
*****
;
; CFG file for writing measurement data
*****
;---- Name of file to be written to startup\ -----
[file]
name          = Messdat.txt

;---- Type of measurement -----
[string]
line          = 1
position      = 1
d             = Radius

;---- Setpoint -----
[string]
line          = 2
position      = 1
d             = Sollwert =

[value]
line          = 2
position      = 20
eparam        = 50
form          = 4.3
sign          = y

;---- Lower tolerance -----
```

```
[string]
line          = 3
position      = 1
d             = Untere Toleranz =
```

```
[value]
line          = 3
position      = 20
eparam        = 51
form          = 4.3
sign          = y
```

```
;--- Upper tolerance -----
```

```
[string]
line          = 4
position      = 1
d             = Obere toleranz =
```

```
[value]
line          = 4
position      = 20
eparam        = 52
form          = 4.3
sign          = y
```

```
;--- Actual value -----
```

```
[string]
line          = 5
position      = 1
d             = Istwert =
```

```
[value]
line          = 5
position      = 20
eparam        = 53
form          = 4.3
sign          = y
```

```
;--- Difference -----
```

```
[string]
line          = 6
position      = 1
d             = Differenz =
```

```
[value]
line          = 6
position      = 20
eparam        = 54
form          = 4.3
sign          = y
```

```
;-----
```

```
[string]
line          = 7
d             = *****
```

7. Tool measuring cycles for laser measuring

7.1 General remarks

AVAILABILITY

The machine and MillPlus must be prepared by the machine manufacturer for the measuring instrument. If not all the G functions described here are available on your machine, consult your machine handbook.

PROGRAMMING

Before calling one of the G600-G609 functions a M24 (active measuring system) must be programmed, so that the measuring system is set in the measuring position.

After measuring a M28 (deactive measuring system) must be programmed, so that the measuring system is retract.

MACHINE CONSTANTS

The G function and associated machine constants are activated via the following machine constants.

MC 261 >0: Measurement cycle functions

MC 254 >0: Measure tool

MC 840 =1: Measuring probe present

MC 854 =1: Tool measuring instrument type (0=none, 1=laser, 2=TT130)

MC 859 =1: Signal type of second stylus (V410 only)

New machine constants

MC 350 Probe position 1st axis negative μm

MC 351 Probe position 1st axis positive μm

MC 352 Probe position 2nd axis negative μm

MC 353 Probe position 2nd axis positive μm

MC 354 Probe position 3rd axis negative μm

MC 355 Probe position 3rd axis positive μm

MC 350 to MC 355 are operator machine constants and are detected when calibrating.

MC 356 Axis number for radial measuring: 1=X, 2=Y, 3=Z

MC 357 Tool axis number for measuring: 1=X, 2=Y, 3=Z

MC 358 Measure: 3rd axis 0=no, 1=yes

MC 359 Radial probe contact side: -1=neg, 0=aut, 1=pos

MC 360 -- MC 369 are intended for a second laser measurement device in another work area or an adapter spindle. The area used is determined by the IPLC.

MC 370 Maximum tool radius μm

MC 371 Maximum tool length μm

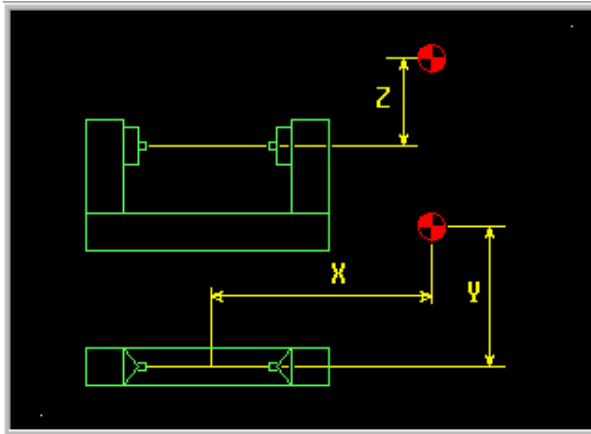
MC 372 Clear space under laser beam μm

MC 373 free space behind the laser beam in μm

7.2 G600 Laser: Calibration

Purpose

To determine the position of the laser measuring instrument and store this position value in the machine constants provided.



Format

N... G600 {X... Y... Z...} {S...} {I1=..}

Parameter

```
G   Laser: Calibration
X   Measure point (optional)
Y   Measure point (optional)
Z   Measure point (optional)
S   Speed (rev/min)
I1= Swing determination 0=no 1=yes
```

Type of function

non-modal

Associated functions

G601, G602, G603, G604

Notes and application

DETERMINING CONCENTRICITY ERROR (I1=)

Use address I1 to specify whether the concentricity error is to be measured and saved in the tool table against the calibration tool. It is recommended that the concentricity error should be determined once using a clean calibration stylus.

- I1= 0 Do not determine concentricity error (basic setting)
 1 Determine the concentricity error

The radial concentricity error is written to the tool memory under R4=.

The axial concentricity error is written to the tool memory under L4= and the length L is reduced by the L4 value. The sum $L+L4$ remains constant.

CALIBRATION STYLUS, TOOL MEMORY ADDRESSES

The dimensions of the calibration mandrel are entered in the tool memory.

L Length of calibration mandrel (underside of cylindrical portion)

R Radius

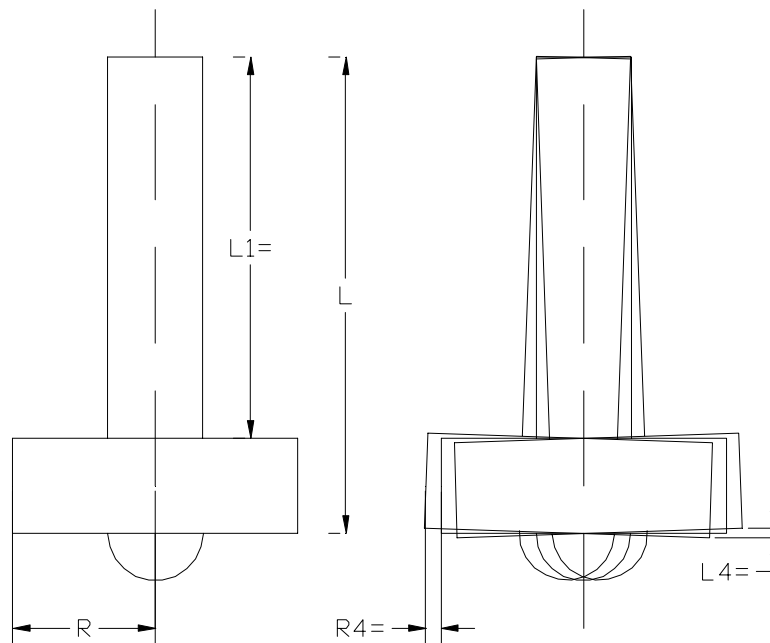
L1= Second length (top of cylindrical portion)

The second length L1= is not entered if a fully cylindrical calibration pin is used. In this case, only the top of the laser beam is calibrated.

The concentricity errors R4 and L4 of the calibration stylus are written to the tool memory by the calibration cycle.

R4= Radial concentricity error of calibration stylus.

L4= Axial concentricity error of calibration stylus.



Definition of calibration tool in tool memory.

POSITION OF MEASURING UNIT

At calibration, the exact position of the measuring instrument is measured and stored in MC350-MC355. The stored values are relative to the reference point of the machine.

When determining the position of the measuring unit for the calibration, the centre of the bottom edge of the pin (dimension L) must be set in the light beam (+/- 5 mm).

X,Y,Z is the global position (to within +/- 5 mm) of the measuring instrument relative to the machine zero point.

If X,Y or Z are not entered, the calibrated positions from the machine constants are used.

- Rotary axes, if any, are not taken into account or positioned.
- Zero point offset must not be active if X,Y or Z are entered.
- The free machining plane G7 and axis rotation G92/G93 B4 must not be active
- A calibration tool must be chosen. T0 is not permitted.

SPEED

S = Speed (recommended value S3000)

Coolant will be thrown off by clockwise-anticlockwise-clockwise rotation.

The spindle is switched off with M5 at the end of the cycle.

Example

Example : Calibrating laser measuring instrument and storing the position value in the E parameter.

N... G600 X300 Y500 Z600 S3000

Example: calibration of laser measuring unit, determining concentricity error.

N... G600 X300 Y500 Z600 I1=1 S3000

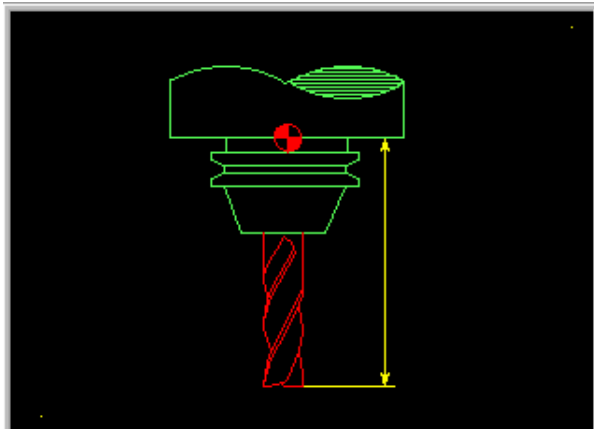
Concentricity errors L4 and R4 are saved in the tool table, length is matched (I1=1).

The exact X, Y and Z positions are saved in the machine constants.

7.3 G601 Laser: Measure tool length

Purpose

To measure the length of centric tools.



Format

N... G601 {S...} {I1=}

Parameter

```
G   Laser: Measure tool length
S   Speed (rev/min)
I1= Measuring side 0=lower,1=upper
```

Type of function

non-modal

Associated functions

G600, G602, G603, G604

Notes and application

SELECTING THE TOOL EDGE (I1=)

The lower edge or the upper edge of the tool can be measured.

- | | | |
|-----|---|------------------------------------|
| I1= | 0 | measure lower edge (basic setting) |
| | 1 | measure upper edge |

SPEED

S = Speed (recommended value S3000)

If the spindle is not first switched off (M5 or M19), then:

- Coolant will be thrown off by clockwise-anticlockwise-clockwise rotation.
- The spindle is switched off with M5 at the end of the cycle.

If the spindle is already switched off (M3 or M4), change of direction or spindle stop does not occur at the end of the cycle

TOOL MEMORY ADDRESSES

The following addresses of the tool memory are used:

- | | |
|-----|---------------------------------------|
| L | Tool length |
| L4= | Allowance length |
| L5= | Length tolerance |
| R6= | Radius position for measuring length. |
| E | Tool status |

ACTIONS

CHECK (E=1):

The measured difference is added to L4 in the tool table.

MEASURE (E=0 or no value):

When the first measurement is made, the tool length is overwritten, and allowance L4=0 and tool status E=1 are set.

- If the tolerance is exceeded, tool status E-1 is set.
- If the tool status is E=1 at the start of the cycle, the cycle is skipped.
- Speed-dependent measurement feed is calculated by the cycle.

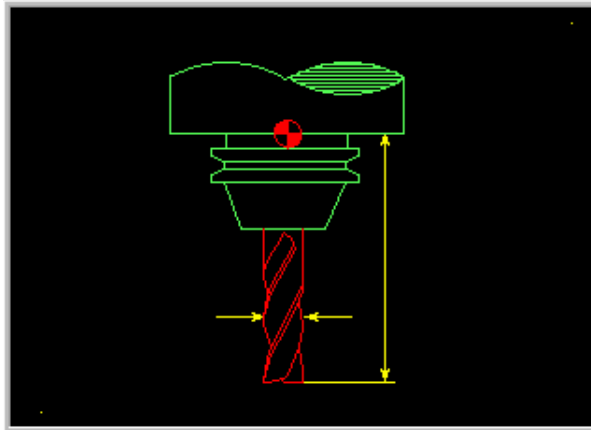
LENGTH MEASUREMENT

- If the tool radius is greater than MC373 and R6 is not programmed, the length is measured eccentrically.
- If R6 is programmed and $R-R6 > MC373$, an error message is issued

7.4 G602 Laser: Measure length and radius

Purpose

To measure the length and radius of acentric tools with laser measuring instrument



Format

N... G602 {S...} {I1=..} {I2=..}

Parameter

```
G  Laser: Measure length and radius
S  Speed (rev/min)
I1= Measuring side 0=lower,1=upper
I2= Double-sided meas. 0=no,1=yes
```

Type of function

non-modal

Associated functions

G600, G601, G603, G604

Notes and application

SELECTING THE TOOL EDGE (I1=)

The lower edge or the upper edge of the tool can be measured.

I1=	0	measure lower edge (basic setting)
	1	measure upper edge

SELECTING MEASUREMENT OF ONE OR BOTH EDGES (I2=)

One or both edges of the tool can be measured.

- I2= 0 measure one side (basic setting)
 1 measure both sides

When measuring both edges, temperature errors and tool obliquity have no influence on the measured radius.

SPEED

S = Speed (recommended value S3000)

If the spindle is not first switched off (M5 or M19), then:

- Coolant will be thrown off by clockwise-anticlockwise-clockwise rotation.
- The spindle is switched off with M5 at the end of the cycle.

If the spindle is already switched off (M3 or M4), change of direction or spindle stop does not occur at the end of the cycle

ADDRESSES OF TOOL MEMORY

The following addresses of the tool memory are used:

- L Tool length
 L4= Length allowance
 L5= Length tolerance
 R Tool radius
 R4= Allowance radius
 R5= Radius tolerance
 L6= Position above the tool tip for true running check
 R6= Radius position for length measurement
 Q4= Number of teeth
 E Tool status

ACTIONS

CHECK (E=1):

The measured deviation is added to L4 and R4 in the tool table.

MEASURE (E=0 or no value):

When the first measurement is made, the tool length and radius are overwritten, and allowance L4 and R4 =0 and tool status E=1 are set.

- If the tolerance is exceeded, tool status E=1 is set.
- If the tool status is E=1 at the start of the cycle, the cycle is skipped.
- Speed-dependent measurement feed is calculated by the cycle.

LENGTH MEASUREMENT

- If the tool radius is greater than MC373 and R6 is not programmed, the length is measured eccentrically.
- If R6 is programmed and $R-R6 > MC373$, an error message is issued.

RADIUS MEASUREMENT

- If L6>0 or not entered (tool table) a radius measurement is carried out.
- If L6 is greater than MC372, an error message is issued.

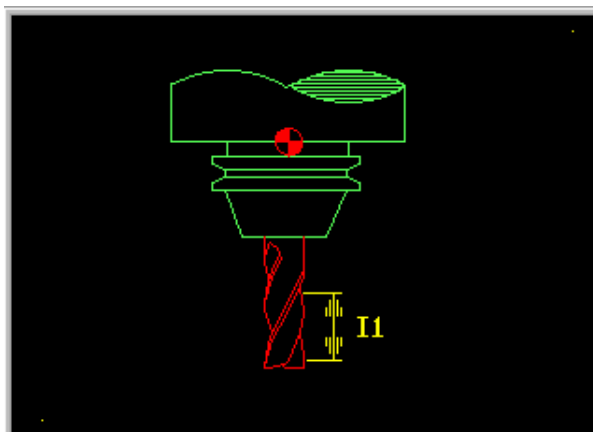
TRUE RUNNING CHECK

- If Q4>0 (number of teeth from the tool table), a true running check is carried out after the radius measurement.
- The true running check is carried out at a calculated rpm.
- The speed superimpose switch is not active.

7.5 G603 Laser: Check of individual edge

Purpose

To monitor the lower part (inspection height) of the tool with a laser measuring instrument.



Format

N... G603 {I1=...} {F2=...}

Parameter

G	Check of individual edge
I1=	Tooltraveldistance (optional)
F2=	Scanning feed

Type of function

non-modal

Associated functions

G600, G601, G602, G604

Notes and application

ADDRESSES OF TOOL MEMORY

The following addresses of the tool memory are used:

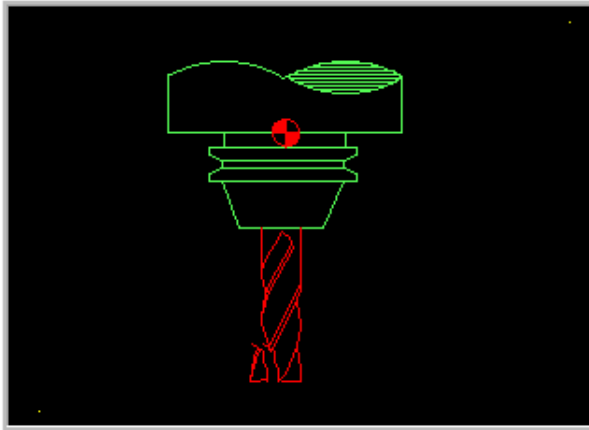
L	Tool length
L4=	Length allowance
R	Tool radius
R4=	Radius allowance
R5=	Radius tolerance
L6=	Position above the tool tip for true running check
Q4=	Number of teeth
E	Tool status

- If the tolerance is exceeded, tool status E-1 is set.
- If the tool status is E=1 at the start of the cycle, the cycle is skipped.
- If I1=0, only a true running check is carried out.
- The edge check is carried out at a calculated speed.
- The speed superimpose switch is not active.
- Maximum error is laid down via R5.
- If I1+L6 is greater than MC372, an error message is issued

7.6 G604 Laser: Tool breakage control

Purpose

Tool breakage check



Format

N... G604 {S...} {I1=..}

Parameter

```
G   Laser: Tool breakage control
S   Speed (rev/min)
I1= Meas. direction 0=pull,1=push
I2= 0=Error/Palette, 1=no error
O1= E-param. number for tool status
```

Type of function

non-modal

Associated functions

G600, G601, G602, G603

Notes and use

MEASURING DIRECTION (I1=)

The measuring direction can be pushing or pulling.

I1= 0 pulling (basic setting)
1 pushing

The fast pulling measurement is preferred, but tools with pronounced concave grinding must be measured pushing, as otherwise the hollow grinding will be detected as a break.

ERROR EVALUATION (I2=)

If a break is detected, various actions can follow:

I2= 0 error message or reject pallet (basic setting)
 1 no error message

If I2=0 is selected, function M105 (tool break detected) is issued in the case of tool breakage. The IPLC switches the laser off and the controller issues an error message.

If, however, a pallet system is present, the pallet is rejected if possible, the current program is interrupted and a new pallet is brought in.

If I2=1 is selected, no error message is issued on tool breakage. Every action must be programmed in the part program. To achieve this, the tool status (value E from the tool memory) can be written directly to an E parameter. See address O1.

TOOL STATUS OUTPUT TO E PARAMETER (O1=)

The tool status (definition E in the tool memory) is written to the specified E parameter. Based on this parameter, the program can determine whether a tool breakage has been detected (status -4). This is only meaningful if the error message has been switched off with I2=1.

SPEED

S = speed (recommended value S3000)

If the spindle is not first switched off (M5 or M19), then:

- Spindle is switched on clockwise (M3).
- The spindle is switched off with M5 at the end of the cycle.

If the spindle is already switched off (M3 or M4), spindle stop does not occur at the end of the cycle.

ADDRESSES OF TOOL MEMORY

The following addresses of the tool memory are used:

L	Tool length
L4=	Length allowance
R	Tool radius
R4=	Radius allowance
B	Breakage tolerance in mm (also in inch mode)
R6=	Radius position for breakage check
E	Tool status

TOOL STATUS

- When the breakage tolerance is exceeded, tool status E-4 is set and in addition an alarm is issued.
- Even if the tool status is E=1 at the start of the cycle, the breakage check is carried out
- The basic setting for tolerance B is entered in MC33. Only 1 or 2 mm is possible. The setting of MC133 is in mm even in inches mode.
- Breakage monitoring must be turned on by means of MC32.

BREAKAGE MEASUREMENT

If the tool radius is greater than MC373 and R6 is not programmed, the length is measured eccentrically.

If R6 is programmed and $R - R6 > MC373$, an error message is issued

8. Measuring system TT130

8.1 General notes

AVAILABILITY

The machine manufacturer for the measuring instrument must prepare the machine and MillPlus. If not all the G functions described here are available on your machine, consult your machine handbook.

PROGRAMMING

Before calling one of the G600-G609 functions a M24 (active measuring system) must be programmed, so that the measuring system is set in the measuring position.

After measuring a M28 (deactivate measuring system) must be programmed, so that the measuring system is retract.

MACHINE CONSTANTS

The G function and associated machine constants are activated via the following machine constants.

MC 261 >0: measurement cycle functions

MC 254 >0: measure tool

MC 840 =1: measurement probe present

MC 854 =1: tool measuring instrument type (0=none, 1=laser, 2=TT130)

New machine constants

MC 350 Probe position 1st axis μm

MC 352 Probe position 2nd axis μm

MC 354 Probe position 3rd axis μm

Coordinates of the TT130 stylus centre point relative to the machine zero point G51 and G53 (-max - +max Φm)

MC 356 axis number for radial measurement: 1=X, 2=Y, 3=Z

MC 357 tool axis number for measuring: 1=X, 2=Y, 3=Z

MC 358 measuring: 3rd axis 0=no, 1=yes

MC 359 radial probe contact side: -1=negative, 0=automatic, 1=positive

MC 360 -- MC 369 are for the second laser measurement system a second work area or an attachment spindle. Which area will be used, is determined by the IPCL.

MC 391 calculation of probe measuring feed

Calculation of probe measuring feed with constant tolerance

Calculation of probe measuring feed with variable tolerance

Calculation of probe measuring feed

MC 392 maximum permitted measurement error for tool measurement with rotating tool
(2 - 1000 Φm)

MC 394 probe measuring feed with tool measurement with non-rotating tool
(10 - 3000 mm/min)

MC 395 distance from tool underside to stylus top for tool radius measurement
(1 - 100000 Φm)

MC 396 diameter or side length of the stylus of the TT130. (1 - 100000 Φm)

MC 397 safety zone around the stylus of the TT130 for pre-positioning. (1 - 10000 Φm)

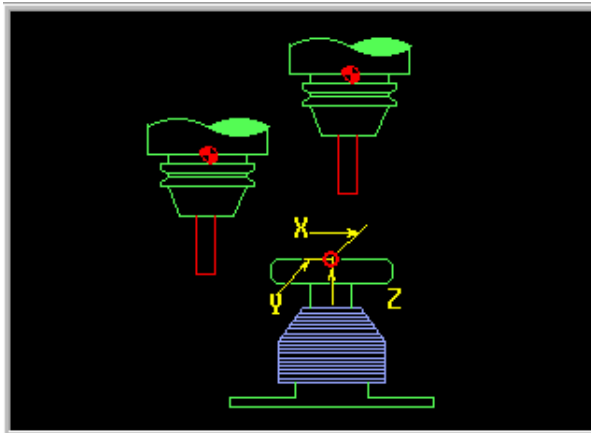
MC 398 rapid in measuring cycle for TT130. (10 - 10000 mm/min)

MC 399 maximum permitted rotational speed at tool edge (1 - 120 m/min)G606

8.2 G606 Calibration TT130

Purpose

To determine the position of the measuring instrument and store this position value in the machine constants provided.



Format

N... G606 {X... Y... Z...}

Parameter

G	TT130: Calibration
X	Measure point (optional)
Y	Measure point (optional)
Z	Measure point (optional)

Type of function

Non-modal

Associated functions

G607, G608, G609

Notes and use

CALIBRATION TOOL

Before you calibrate, you must enter the exact radius and the exact length of the calibration tool in the tool table.

SEQUENCE

The calibration process runs automatically. MILLPLUS also determines the centre offset of the calibration tool automatically. For this, MILLPLUS rotates the spindle after half of the calibration cycle by 180°. As a calibration tool, use an exactly cylindrical part, e.g. a cylindrical pin. MILLPLUS stores the calibration values in the machine constants and takes them into account in the subsequent tool measurements.

In MC 350, MC 352, MC 354 the position of the TT130 in the work area of the machine must be stipulated.

If you change one of MC 350, MC 352, MC 354, you must re-calibrate.

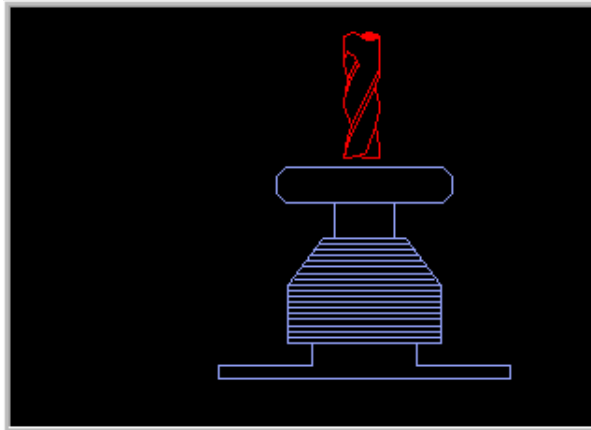
POSITION

Input in the X, Y and Z-axes, the position in which the possibility of collision with workpieces or clamping fixtures is excluded. If the position height input is so small that the tip of the tool would be below the plate surface, MILLPLUS positions the calibration tool above the plate non-automatically.

8.3 G607 Measuring tool length TT130

Purpose

To measure the tool length.



Format

N... G607 {I1=...} {I2=...}

Parameter

```
G    TT130: Measure tool length
I1=  Clearance                (optional)
I2=  All teeth                (0=none, 1=all)
```

Type of function

Non-modal

Associated functions

G606, G608, G609

Notes and use

TOOL DATA

Before you measure tools for the first time, enter the estimated radius, estimated length, number of edges and cutting direction of the tool in each case into the tool table.

ADDRESSES OF THE TOOL MEMORY

The following addresses of the tool memory are used:

L	tool length
L4=	length allowance
L5=	length wear tolerance
R	tool radius
R4=	radius allowance
R6=	measurement offset radius
E	tool status

SEQUENCE

The tool length can be determined in three different ways:

- 1 If the tool diameter is greater than the diameter of the measurement surface of the TT130, measure with tool rotating.
- 2 If the tool diameter is smaller than the diameter of the measurement surface of the TT130 or if you determine the length of drills or radius cutters, measure with tool stationary.
- 3 With the parameter 'All teeth' all teeth are measured. The measurement is carried out with stationary spindle. The greatest tooth length is entered in the tool table.

MEASURING WITH TOOL ROTATING

To determine the longest edge, the tool to be measured is offset to the probe system centre point and moved, rotating, onto the measurement surface of the TT130. Program the offset in the tool table under tool offset; radius (R).

MEASURING WITH TOOL STATIONARY (e.g. for drills).

The tool to be measured is moved to be concentrically above the measurement surface. Then it travels with the spindle stationary onto the measurement surface of the TT130. For this measurement enter the tool offset: radius (R6=0) in the tool table.

INDIVIDUAL EDGE MEASUREMENT

MILLPLUS pre-positions the tool to be measured to the side of the probe. The end face of the tool is then located below the probe top as laid down in MC 395. In the tool table, you can stipulate an additional offset under tool offset; length (L). MILLPLUS applies the probe radial with the tool rotating, to determine the start angle for the individual edge measurement. It then measures the length of all edges by changing the spindle orientation. For this measurement, select the Softkey all teeth.

MEASURE TOOL (E=0 or no value)

Establish whether the tool is being measured for the first time or whether you would like to check a previously measured tool. For the first measurement, MILLPLUS overwrite the tool length L in the tool memory and sets the allowance L4 = 0.

Before you measure tools for the first time, enter the estimated radius (R10), estimated length (L100), number of edges (Q4=4) and the cutting direction (I2=0) of the tool in each case in the tool table.

With the first measurement, MILLPLUS overwrite the tool radius (R10 with R10.012) and tool length (L100 with L 99.456) in the tool memory and sets the allowance R4 and L4 = 0.

CHECK TOOL (E=1)

If you check a tool, the measured length is compared with the tool length L from the tool table. MILLPLUS calculates the difference with the correct sign and enters this as allowance L4 in the tool table. If the allowance is greater than the permitted wear (L5=) or breakage tolerance for the tool length, it gives an error message.

If you check a tool, the measured tool data are compared with the tool data from the tool table. MILLPLUS calculates the differences with correct signs and enters these as the allowance (R4=0.015 and L4=0.06) in the tool table.

SAFE HEIGHT

Input the position in the spindle axis by means of parameters (I1= safety distance) from the dialog input in which the possibility of a collision with workpieces or clamping fixtures is excluded. The safe height relates to the top of the measuring device. If the safe height input is so small that the tip of the tool would lie below the plate surface, MILLPLUS positions the tool non-automatically above the plate (safety zone from MC 397).

EDGE MEASUREMENT

With parameter I2=1, all teeth are measured.

With I2=0 or no value, individual edge measurement is carried out.

STATIONARY SPINDLE

MILLPLUS uses the probe measuring feed from MC 394 for the measurement with stationary spindle.

CALCULATION OF THE SPINDLE Speed

When measuring with a tool, MILLPLUS calculates the spindle speed and the probe measuring feed automatically. The spindle speed is then calculated as follows:

$$n = \frac{\text{MC 399}}{r * 0.0063}$$

where:

n	= speed (rpm)
MC 399	= maximum permitted rotational speed [m/min]
R	= active tool radius [mm]

CALCULATING PROBE MEASURING FEED

The probe measuring feed is calculated from:

$$V = \text{measurement tolerance} * n$$

where:

V	= probe measuring feed [mm/min]
Measurement tolerance	= measurement tolerance [mm], dependent on MC 391
n	= speed [1/min]

METHOD CALCULATING PROBE MEASURING FEED

With MC 391 you adjust the calculation of the probe measuring feed:

MC 391=0: The measurement tolerance remains constant - independently of the tool radius. With very large tools, the probe measuring feed is reduced to zero, however. The smaller the maximum rotational speed (MC 399) and the permitted tolerance (MC 392) you select, the sooner this effect becomes noticeable.

MC 391=1: The measurement tolerance changes as the tool radius increases. This still ensures sufficient probe measuring feed even with large tool radii. MILLPLUS changes the measurement tolerance in accordance with the following table:

Tool radius	measurement tolerance
Up to 30 mm	MC 392
30 to 60 mm	2 * MC 392
60 to 90 mm	3 * MC 392
90 to 120 mm	4 * MC 392

MC 391=2: The probe measuring feed remains constant, though the measurement error increases linearly as the tool radius becomes greater:

$$\text{Measurement tolerance} = \frac{R * \text{MC 392}}{5 \text{ mm}}$$

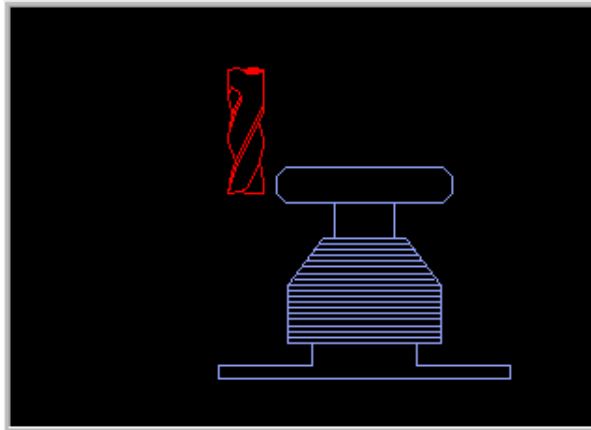
Where:

r = tool radius [mm]
MC 392 = maximum permitted measurement error

8.4 G608 Measuring tool radius TT130

Purpose

To measure the tool radius.



Format

N... G608 {I1=...} {I2=...}

Parameter

```
G    TT130: Measure tool radius
I1=  Clearance                (optional)
I2=  All teeth                (0=none, 1=all)
```

Type of function

Non-modal

Associated functions

G606, G607, G609

Notes and use**ADDRESSES OF THE TOOL MEMORY**

The following addresses of the tool memory are used:

L	tool length
L4=	length allowance
R	tool radius
R4=	radius allowance
R5=	radius wear tolerance
E	tool status

TOOL DATA

Before you measure tools for the first time, enter the estimated radius, estimated length, number of edges and cutting direction of the tool in each case into the tool table.

MEASURE TOOL (E=0 or no value)

With the first measurement, MILLPLUS overwrite the tool radius R in the tool memory and sets the allowance R4 = 0.

MEASUREMENT SEQUENCE

You can determine the tool radius in two ways:

- 1) Measurement with rotating tool
- 2) Measurement with rotating tool and subsequent individual edge measurement

With individual edge measurement, the radius is first measured roughly and the position of the largest tooth determined. After that, the other teeth are measured.

MILLPLUS pre-positions the tool to be measured to the side of the probe. The milling cutter end face is then below the top of the probe, as laid down in MC 395. MILLPLUS applies probe measuring radial with rotating tool. If an individual edge measurement is also to be carried out, the radii of all edges are measured by means of spindle orientation.

CHECK TOOL (E=1)

If you check a tool, the measured radius is compared with the tool radius R from the tool table. MILLPLUS calculates the difference with correct sign and enters this as allowance R4 in the tool table. If the allowance is greater than the permitted wear (R5=) or breakage tolerance for the tool radius, an error message is output.

SAFE HEIGHT

Input the position in the spindle axis by means of parameters (I1 = safety distance) from the dialog input, at which the possibility of a collision with workpieces or clamping fixtures is excluded. The safe height is relative to the active workpiece reference point. If the safe height input is so small that the tip of the tool would be below the top of the plate, MILLPLUS positions the tool non-automatically above the plate (safety zone from MC 397).

EDGE MEASUREMENT

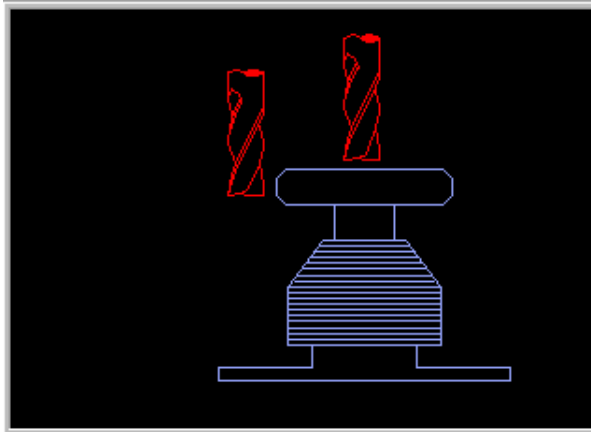
With parameter I2=1 all teeth are measured.

With I2=0 or no value, an individual edge measurement is carried out.

8.5 G609 Measuring length and radius TT130

Purpose

To measure tool length and radius.



Format

N... G609 {I1=...} {I2=...}

Parameter

```
G    TT130: Measure length and radius
I1=  Clearance                (optional)
I2=  All teeth                 (0=none, 1=all)
```

Type of function

Non-modal

Associated functions

G606, G607, G608

Notes and use**ADDRESSES OF THE TOOL MEMORY**

The following tool memory addresses are used:

L	tool length
L4=	length allowance
L5=	length wear tolerance
R	tool radius
R4=	radius allowance
R5=	radius wear tolerance
E	tool status

TOOL DATA

Before you measure tools for the first time, enter the estimated radius, estimated length, number of edges and cutting direction of the tool in each case in the tool table.

MEASUREMENT SEQUENCE

MILLPLUS measures the tool according to a fixed, programmed sequence. First the tool radius and then the tool length are measured.

MEASURE TOOL (E=0 or no value)

The function is especially suitable for the first measurement of tools since, compared with the individual measurement of length and radius, there is a considerable time advantage.

With the first measurement, MILLPLUS overwrite the tool radius R and tool length L in the tool memory and sets the allowance R4 and L4 = 0.

CHECK TOOL (E=1)

If you check a tool, the measured tool data are compared with the tool data from the tool table. MILLPLUS calculates the differences with correct signs and enters these as allowance R4 and L4 in the tool table. If an allowance is greater than the permitted wear (L5= and R5=) or breakage tolerance for the tool radius, an error message is output.

SAFE HEIGHT

Input the position in the spindle axis by means of parameters (I1 = safety distance) from the dialog input, at which the possibility of a collision with workpieces or clamping fixtures is excluded. The safe height is relative to the active workpiece reference point. If the safe height input is so small that the tip of the tool would be below the top of the plate, MILLPLUS positions the tool non-automatically above the plate (safety zone from MC 397).

EDGE MEASUREMENT

With parameter I2=1 all teeth are measured.

With I2=0 or no value, an individual edge measurement is carried out.

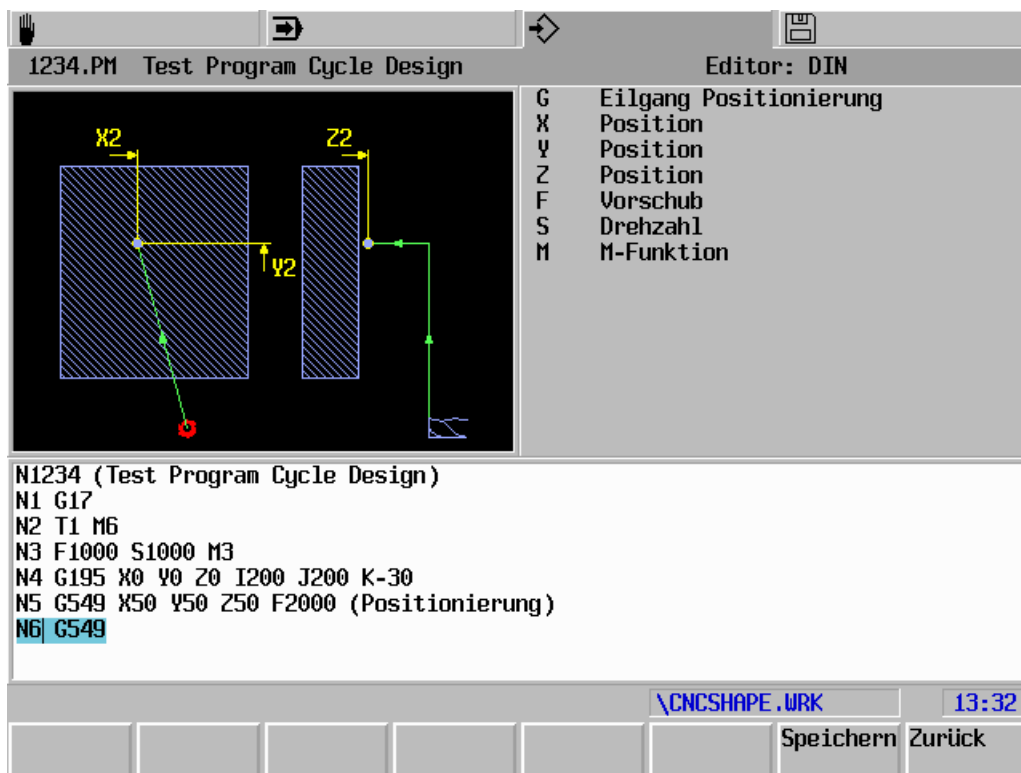
9. Cycle Design

9.1 Introduction

Cycle Design allows the user to define his own G functions and integrate them into the control. These G functions can be programmed within partprograms using graphics support.

The defined G function can be seen in the control in three places.

- 1) The G function list is enlarged by the new G function.
- 2) When programming, the address list is displayed with support graphics.
- 3) The G code can be seen in the partprogram.



For a G function, the following files are required:

- 1) The G function with address information and texts. File G5???.CFG
(5?? is the three-digit number of the G code)
(See paragraph .2: Description of G function and addresses)
- 2) The processing macro (9995???.mm) which is called up by the G function.
(See paragraph .4: execution macro)

The following files are used as required:

- 3) Up to 10 support graphics. File *.DXF or *.BMP
(See paragraph .3: support graphics)
- 4) Up to 9 auxiliary macros (9995??*.mm) which are called up by the processing macro.
(* is a serial number, one digit).
- 5) The general auxiliary macros (99905**.mm) can be called up by several processing macros. (** is a serial number, two digits).

A maximum of 50 G functions can be defined in the range G500 to G599.

Note: In version 410 the following G functions are permitted:
G540 to G563, G570 to G580 and G585 to G599.

The files for each G function are stored on the MILLPLUS hard disk in a dedicated directory D:\STARTUP\CYCLES\G5??.

(See paragraph .5: reading in cycle files).

Users themselves must create these directories.

Example:

The files for G550 and G560 are stored in the directory D:\STARTUP\CYCLES\G550\ and ...G560\.
Two general auxiliary macros are stored in the separate directory D:\STARTUP\.

D:\STARTUP\CYCLES \G550\	G550.CFG	< Graphic 1>.DXF
		< Graphic 2>.BMP
		9995500.mm
		9995501.mm
	\G560\ G560.CFG	< Graphic 1>.DXF
		< Graphic 2>.DXF
		9995600.mm
		9995601.mm
D:\STARTUP\	1234567.mm (Maximal 7 digits	
	*.mm	

9.2 Description of G function and addresses (G5?? CFG)

Each G function and associated addresses must be described in a file (G5?? CFG).

The file defines:

- 1) The number of the G function via the file names.
- 2) The explanatory text for the G function in several languages.
- 3) For each address, a parameter list that includes: address name, E parameters, format, sign, minimum, maximum and explanatory text.
- 4) The file names of up to 10 support graphics.

9.2.1 Example- G5?? CFG file (definition G5?? CFG)

```
[DIALOG]           ; must be present.
GB                 =      Positioning
D                  =      Positionierung
NL                 =      Positionering
```

; Comments

```
[PARAMETER]
ADDRESS           =      X2
EPARAM            =      260
OPTIONAL          =      N
FORM              =      6. 3
SIGN              =      Y
INCH              =      Y
MIN               =      -999999.999
MAX               =      999999.999
GB                =      Position
D                 =      Position
NL                =      Positie
```

```
[PARAMETER]
ADDRESS           =      F
EPARAM            =      252
OPTIONAL          =      Y
FORM              =      5.0
SIGN              =      N
INCH              =      Y
MIN               =      0
MAX               =      99999
GB                =      Feed
D                 =      Vorschub
NL                =      Voeding
```

```
[SUPPORT]
PIC01             =      *.DXF ;file names of first support graphic
...
PIC10             =      *.BMP ;up to 10 support graphics
```

Remarks:

- 1) Blanks (except in texts), tabs and blank lines have no meaning.
- 2) Comments (text after ";") are permitted anywhere.
- 3) Keywords: [DIALOG], [PARAMETER] and [SUPPORT]. [PARAMETER] is to be defined for each address. The keyword [DIALOG] must be entered. Optional are [PARAMETER] and [SUPPORT]. After the keywords, information may be entered.
- 4) After [DIALOG] G function texts in several languages may be entered.

<Language code> = For each language an explanatory text (up to 33 characters) may be entered (default setting no text)
Only the languages required need be entered.

The actual choice of language is determined by machine constant MC5.

Language codes are:

GB, D, NL, F, I, E, DK, S, SF, P, PL, CZ

- 5) After [PARAMETER] the address data follow.

A maximum of 25 addresses are permitted.

ADDRESS = Name of the address. Available address names see paragraph 2.3 (permitted addresses). Also E0 to E400 is a valid name.
All other names produce error message O141.

If ADDRESS is missing, no address is defined.

EPARAM = E parameter number which is assigned to the address value entered. Range 0 up to maximum number of parameters (MC 83).
For Example: EPARAM=260 and ADDRESS=X2 means that E260 contains the programmed value of X2. If X2 is not programmed in the workpiece program, E260 is set as equal to-999999999.

If ADDRESS=E***, the same number must be entered after EPARAM=***.

If EPARAM is absent, no address is defined.

FORM = Determines the input format (default 6.3). 6.3 means: 6 figures before the decimal point and 3 after. Max. format in mm is 6.3 and in inches 5.4.

SIGN = [Y/N] Determines whether the input can be negative or positive. (Default Y).

MIN = Smallest input (default -999999999).

MAX = Largest input (default 999999999)

OPTIONAL = [Y/N] Determinate of the input is obligatory (default is Y).
Optional Addresses are distinguished in colour (white) in the display.
If an obligatory address (not optional) is not given, the process will be stopped and an error message will be given. (from V420).
In macro must be defined a default for an optional address.

ACTIVE = [1, 2, --, 9] Stands for a coupling between graphics and addresses. The address is only shown, when the concerning graphic is chosen.
(Default: active by all graphics) (From V420)

<Language code> = For each language an explanatory text (up to 33 characters) may be entered (default setting no text)
Only the languages required need to be entered.
The actual choice of language is determined by machine constant MC5.
Language codes see [DIALOG].

- 6) After [SUPPORT] follow the file names of a maximum of 10 support graphics in *.DXF or *.BMP format. Valid file names are up to 8 characters in length, ending in *.DXF or *.BMP.
PIC number determines the sequence of the graphics in the control.

PIC01 = Defines the file name of the first support graphic.
(Default no support graphic).

PIC02 = Defines the file name of the second support graphic.

And so on.

- 7) Changes to the G5???.CFG file will be only active after power down the machine.

9.2.2 Example-G550.CFG file

;Comments...

[DIALOG]

GB = Positioning
D = Positionierung

[PARAMETER]

ADDRESS = X
EPARAM = 260
OPTIONAL = Y
FORM = 6.3
SIGN = Y
MIN = -999999.999
MAX = 999999.999
GB = Position
D = Position

[PARAMETER]

ADDRESS = Y
EPARAM = 261
OPTIONAL = Y
FORM = 6.3
SIGN = Y
MIN = -999999.999
MAX = 999999.999
GB = Position
D = Position

[PARAMETER]

ADDRESS = E1
EPARAM = 1
OPTIONAL = Y
FORM = 6.3

CYCLE DESIGN

SIGN = Y
 MIN = -999999.999
 MAX = 999999.999
 GB = Position
 D = Position

[SUPPORT]
 PIC01 = ABCD1234.DXF

9.2.3 Permitted addresses

In **CYCLE DESIGN** only the addresses below are permitted.

A	A1=	A2=	A3=	A40=	A4=	A5=	A6=	A7=	A90=	A91=		
B	B1=	B2=	B3=	B40=	B47=	B4=	B5=	B6=	B7=	B90=	B91=	
C	C1=	C2=	C3=	C4=	C40=	C5=	C6=	C7=	C90=	C91=		
D	D1=	D2=	D3=	D4=								
E	ENNN=											
F	F1=	F2=	F3=	F4=	F5=	F6=						
G	G1=											
H												
I	I1=	I2=	I3=	I4=	I5=	I6=	I7=	I90=	I91=			
J	J1=	J2=	J90=	J91=								
K	K1=	K2=	K3=	K4=	K5=	K90=	K91=					
L	L1=	L2=	L3=	L4=	L5=	L6=	L7=					
M	M1=	M2=										
N	N1=	N2=	N=									
O	O1=	O2=										
P	P1=	P2=	P3=	P4=								
Q	Q1=	Q2=	Q3=	Q4=								
R	R1=	R2=										
S	S1=	S7=										
T	T1=	T2=	T3=									
U	U1=	U2=	U7=									
V	V1=	V7=										
W	W1=	W2=	W3=	W4=	W5=	W7=						
X	X1=	X2=	X3=	X4=	X51=	X52=	X53=	X61=	X62=	X7=	X90=	X91=
Y	Y1=	Y2=	Y3=	Y4=	Y51=	Y52=	Y53=	Y61=	Y62=	Y7=	Y90=	Y91=
Z	Z1=	Z2=	Z3=	Z4=	Z51=	Z52=	Z53=	Z61=	Z62=	Z7=	Z90=	Z91=

Until the V420 only the fat printed addresses are permitted.

9.3 Support graphics

For the CNC, the graphics are to be read in a standard format (*.DXF or *.BMP).

Usually, a drawing program is used that can generate a *.DMF format from Autocad9 to Autocad14.

This *.DXF or *.BMP file must be transmitted to the control. It is then automatically converted in the control into an internal *.PIC file.

9.3.1 Making graphics in *.DXF format

When using AutoCAD, the following rules must be observed:

- Define LIMIT [331,240]. This is the dimension in PIXELS of the support graphic.
- The graphic is to be centred in the middle.
- Define POINT [0,0] as the corner of the drawing
- Define POINT [331,240] as the second corner of the drawing
- The following are supported: LINE, POINT, CIRCLE, MARK, SOLID, TEXT, POLYLINE, LINETYPE and COLOR
- For geometry elements, the delta-X and/or delta-Y must not be greater than 255.
- A radius must not be greater than 127.
- Auxiliary drawings (ELEMENTS) must be inserted (INSERT), then perform PURGE and EXPLODE.
- Polyline must not be longer than 40 points.
- Hatching must be taken into the drawing with EXPLODE.
- Finish drawings with ZOOM and EXTEND.

A *.DMF file is generated with the command DXFOUT.

Note: "AutoCAD" is a registered product.

9.3.2 Making graphics in *.BMP format

When using Bitmap, the following rules must be observed:

- The dimension in PIXELS of the support graphic must be 331 by 240.
- The graphic is to be centred in the middle.
- The file size must be smaller than 64K.
- The format must be compressed.

9.3.3 Colour code

The available colours are:

1	red	9	pale red
2	yellow	10	pale yellow
3	green	11	pale green
4	cyan	12	pale cyan
5	blue	13	pale blue
6	dark grey	14	magenta
7	grey	15	white
8	black	16	black

9.3.4 Types of line

The available types of line are:

- 1 : _____
- 2 : _____
- 3 : _____
- 4 : _____
- 5 : _____
- 6 : _____
- 7 : _____
- 8 : _____

9.4 Execution macro

When executing the G function the macro with the desired sequence is started.

This macro can be written directly in DIN/ISO (*.MM).

Example of execution macro

```

N9995500 (positioning)
N1 (description of E parameters used)
N2 (E150 = jump parameter]
N3 (E252 = Feed]
N4 (E254 = X_tool position)
N5 (E255 = Y_tool position)
N6 (E256 = Z_tool position)
N7 (E260 = X2_end position)
N8 (E261 = Y2_end position)
N9 (E262 = Z2_end position)
N1000 G29 I1 E150 N=1002 E150=E252=-999999999
                (Jump to N1002 if F is not programmed i.e. equal to  999999999)
N1001 F=E252                (activate feed F)
N1002 G326 X7=254 Y7=255 Z7=256                (read current position)
N1003 G29 I1 E150 N=1005 E150=E260>-999999999
                (Jump to N1005 if X2 is programmed, i.e. greater than  999999999)
N1004 E260=E254                (X2 is set equal to the current position)
N1005 G29 I1 E150 N=1007 E150=E261>-999999999
                (Jump to N1007 if Y2 is programmed, i.e. greater than  999999999)
N1006 E261=E255                (Y2 is set equal to the current position)
N1007 G29 I1 E150 N=2030 E150=E262>-999999999
                (Jump to N2030 if Z2 is programmed, i.e. greater than  999999999)
N1008 E262=E256                (Y2 is set equal to the current position)
N2030 G1 X=E260 Y=E261 Z=E262                (in feed to X2, Y2, Z2)

```

9.4.1 Explanation

- 1) For the meanings of the G-functions see control operating instructions.
- 2) The free E parameters to be used by Cycle Design are in the range from E0 to MC83 maximum.
- 3) The programmed addresses of the cycle are transferred in E parameters from the G550.CFG file. Addresses not programmed are set equal to - 999999999. It is thus possible to recognise whether an address is programmed. This also applies to addresses such as F, which are otherwise modal.
- 4) In this example, all addresses are optional. When programming, control checks whether the value meets the criteria set from file G550.CFG. In the macro, there must be a reaction if the value is not entered. In this example, for a non-programmed Z2 address the current position is taken.
- 5) Inputs, which are mutually exclusive must be recognised in the macro. The macro may issue an error message such as P07 (value outside range).
- 6) Changes to the *.MM file, will be only active after power down the machine.

9.5 Reading cycle files into the CNC

*.DXF, *.BMP, *.CFG and *.MM files can be loaded into the control via an Ethernet link.

*.mm files can be loaded directly from a PC into the control with a suitable data communications program, such as CDS.

For example, G550.cfg is to be loaded in the file directory D:\STARTUP\CYCLES\G550. Each G function has its own file directory.

Inside the CNC the text files can be produced and edited with the hard disk editor.

Changes made in the file with the editor, will be only activated after the machine switching on and off

10. Technological instructions

10.1 F-Function

Purpose

To set the feed in millimetres per minute or per revolution (mm/min or mm/rev). The feed rate actually used depends on several factors, for instance material, type of machining and tool.

Format

N... {F..} {F1=..} {F2=..} {F3=..} {F4=..} {F5=..} {F6=..}

Definitions, abbreviations:

F: General feed for axis movements with G1/2/3
 F1=: Selection of constant cutting feed rate for radius compensation of circles.
 F2=: Retract feed at G85, infeed at G87/G89, G201 or measuring feed at G145.
 F3=: Feed for the (negative) infeed movement (infeed).
 F4=: Feed for the plane movement
 F5=: Feed unit for rotary axes
 F6=: Local feed

Associated functions

G1, G2, G3, G41/G42, G45, G61/G62, G70/G71, G81/G89, G94/G95, G145, G201

Type of function

Modal	F, F1=, F3=, F4=, F5=
Blockwise	F2=, F6=

Notes and usage

The feed often has to be changed for technological reasons. The essential factors for making adjustments are:

- 1) Feed and direction of the movement
- 2) Constant cutting feed rate for radius compensation of circles
- 3) Feed in cycles
- 4) Feed unit for rotary axes
- 5) Local feed

1) Feed and direction of the movement:

During cutting operations the feed should be carefully matched with the milling process for technological reasons. Technological conditions for milling in radial direction are different from those in axial direction.

It is most advantageous to the user if he is able to program 2 feed values modally and independently. Independent feed programming is possible with parameters F3= and F4=.

Feeds F, F3= and F4= are modal and programmable:

(0 ... 99999 [mm/min] metric)

(0 ... 9999.9 [inch/min] inch)

F3=: Sets the infeed

F4=: Sets the plane feed

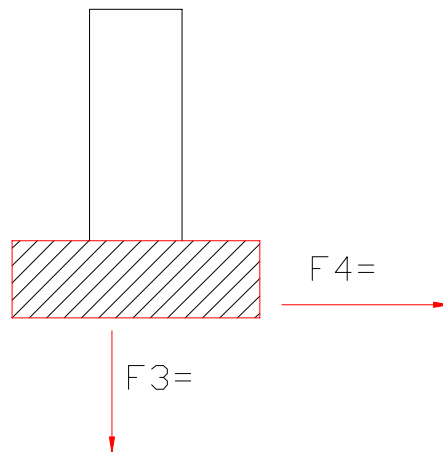
F: Sets the infeed and plane feed

If F, F3= and F4= are programmed in a block, F3= and F4= have a higher priority than F.

Tool axis: Axis perpendicular to the machining plane (G17, G18, ...).

Radial milling direction: Milling in the machining plane

Axial milling direction: Milling into the direction of the tool axis (in infeed direction only)



F4= Radial milling direction

F3= Axial milling direction

- Infeed: only effective in the blocks dealing with infeed movements only.
- Plane feed: effective for all other movements not involving pure infeed movements.

Initialisation: F3=0, F4=0 and F = 0

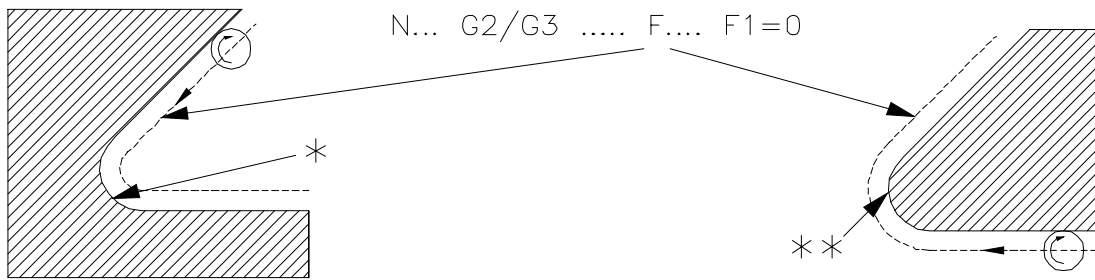
After M30, CANCEL PROGRAM Softkey or CLEAR CONTROL Softkey, F, F3= and F4= are zeroed.

2) Constant cutting feed by radiuskompensation of circles

The parameter 'F1=.' is used to ensure that the programmed feed rate along a workpiece contour remains constant regardless of the radius of the mill and the contour shape. This controlled velocity is called the CONSTANT CUTTING FEED.

F1=0 CONSTANT CUTTING FEED not applied (default mode; also set at CLEAR CONTROL or M30 or Softkey CANCEL PROGRAM).

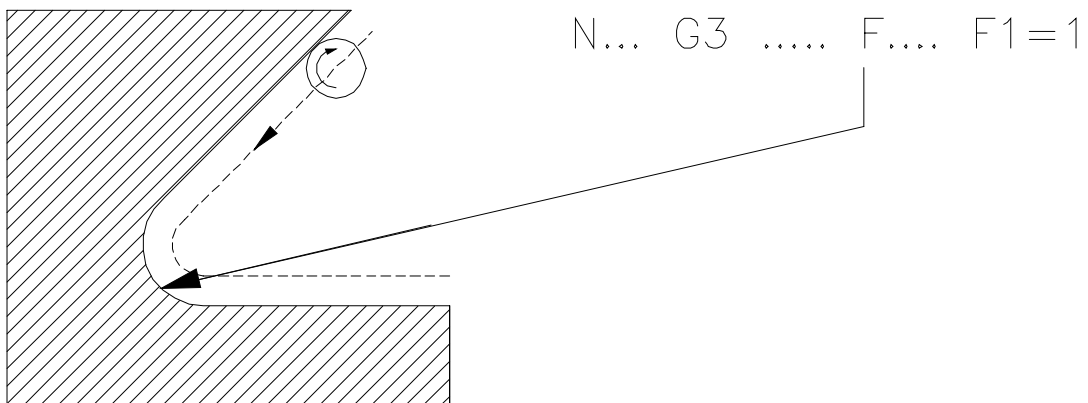
The programmed feed rate should be the velocity of the tooltip.



* Cutting feed too high

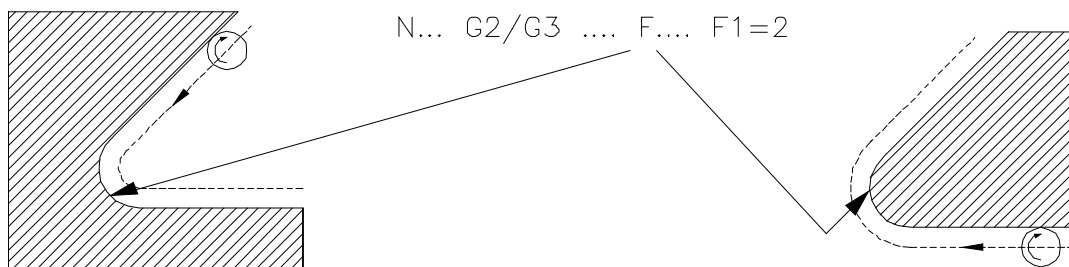
** Cutting feed too low

F1=1 CONSTANT CUTTING FEED applied only on the inside of arcs. The programmed feed rate is reduced to assure that the tooltip moves with the reduced velocity on the inside of an arc.



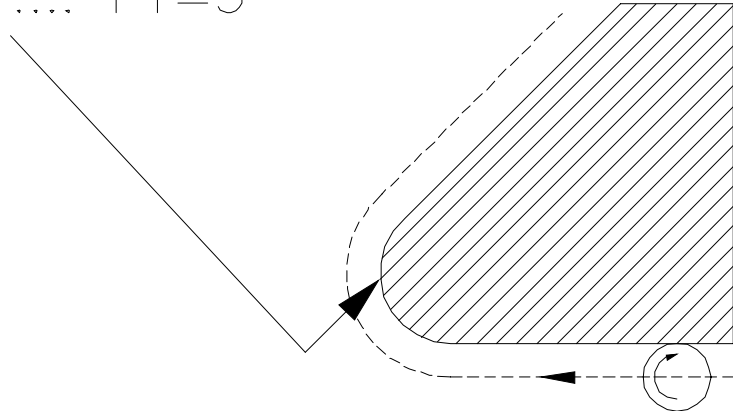
NF

F1=2 CONSTANT CUTTING FEED applied on the inside and outside of arcs. The programmed feed rate is reduced (inside arc) or increased (outside arc) to assure that the tooltip moves with the recalculated velocity. If the increased velocity is greater than the maximum feed rate (a Machine Constant value) the maximum feed rate is used.



F1=3 C.C.F. applied only on the outside of arcs. The programmed feed rate is increased to assure that the tooltip moves with the increased velocity on the outside of an area. If the increased velocity is greater than the maximum feed rate (a Machine Constant value) the maximum feed rate is used.

N... G2 F.... F1=3



3) Feed in cycles

In cycles G81, G83, G85 and G86 the movement in "axial" direction is not an infeed movement, but a feed movement. It is therefore programmed with F/F4=, not with infeed F3=.

In cycles G87, G88 and G89 the infeed movement can be programmed block-by-block using F2= and modally with F3=.

F3= is used as infeed in Easy Operate cycles.

Messages

- If feeds are missing (e.g. at F3=0 or F4=0 or F0).
Message: P04 No feed is programmed

4) Feed unit for rotary axes

G94 F5=: feed of the rotary axes
F5=0 degrees/min (default)
F5=1 mm/min or inches/min)

MACHINES WITH KINEMATIC MODEL

Function G94 F5= is only possible if a kinematics model is defined for the machine. (MC312 must be active).

ROTARY AXIS RADIUS CALCULATION G94 F5=1

In machines with the kinematics model, the rotation axis radius between the centre point of the rotary axis and workpiece can be calculated. Because of this, A40=, B40= and C40= no longer need to be programmed. The new possibility is programmed via G94 F5=1.

SHUT DOWN G94 F5=1

G94 F5=1 is cancelled by G94 F5=0, G95, programming with A40=, B40= or C40= in G0 or G1, M30, <abort program> or <reset CNC>.

5) Local feed

F6= is a local feed which is only active in the record in which it is programmed. F is the normal feed and also applies to the following records.

MAXIMUM FEED

The maximum feed with which the machine tool may be operated is stated in the machine documents (MC740).

RAPID MOVEMENT

A F6= in a G0 block limited the feed. The movement will be a rapid movement. That means for example that the programming logic stays active.

Example

N10 F1000

N10 : Set feed to 1000 mm/min

10.2 H Auxiliary function

Purpose

Only the machine builder may use the H-function. Refer to the machine documentation.

Format

N... H...

Type of function

Depending on machine adaptation component.

Notes and usage

MAXIMUM NUMBER OF DECIMAL PLACES

The H-function can have a maximum of four decimal places.

10.3 M-Funktionen

10.3.1 M0 Program stop

Purpose

To interrupt the execution of a program.

Format

N... {Programmed tool movement} M0

Associated functions

M1

Type of function

Non-modal

Notes and usage

ACTIVATION

The M0 function will become active when the current tool movement programmed in the same block has been executed.

SPINDLE SPEED AND COOLANT SUPPLY

The machine tool interface determines whether the spindle rotation and coolant are suppressed or cancelled as well.

Suppressed means that the spindle starts rotating and the coolant supply is switched on after resuming the execution of the partprogram.

Cancelled means that a spindle stop is executed and the coolant supply switched off. These functions have to be programmed again after the stop command.

RESUME PROGRAM EXECUTION

The execution of the program is resumed after the START button has been pressed.

Example

N200 G1 X100 Y100 F200 M0

N200: Move the tool to the programmed position and then halt the execution of the program.

10.3.2 M1 Optional program stop

Purpose

To interrupt the execution of a program if this function is encountered and the Softkey OPTIONAL STOP in MACHINING is activated.

Format

N... {Programmed tool movement} M1

Associated functions

M0

Type of function

Non-modal

Notes and usage

ACTIVATION

The M1 function will become active when the current tool movement programmed in the same block has been executed.

SPINDLE SPEED AND COOLANT SUPPLY

The machine tool interface determines whether the spindle rotation and coolant are suppressed or cancelled as well.

Suppressed means that the spindle starts rotating and the coolant supply is switched on after resuming the execution of the program.

Cancelled means that a spindle stop is executed and the coolant supply switched off. These functions have to be programmed again after the stop command.

RESUME PROGRAM EXECUTION

The execution of the program is resumed after the start button has been pressed.

Note: If a M1, optional stop, is programmed and optional stop mode is not active then the following G1 blocks are executed with inpos instead of inpos disregarding the G28 function. In order to get correct behaviour the programmer should program a machine function in the block next to the M1-block.

Example

N200 G1 X100 Y100 F200 M1

N200 Move tool to programmed position and then halt program execution, if the Softkey OPTIONAL STOP is active.

10.3.3 M3/M4 Spindle-rotating clockwise/counter clockwise

Purpose

To switch on spindle rotation in clockwise (CW) or counter clockwise (CCW) direction.

- M3 Spindle rotation in clockwise direction.
- M4 Spindle rotation in counter clockwise direction.

Format

N... {Programmed tool movement} M3/M4

Associated functions

- M5, M19
- M41, M42, M43, M44

Type of function

Modal

Notes and usage

START SPINDLE ROTATION

Spindle starts rotating before the tool movement programmed in the same block has been executed.

CANCELLATION

The direction of spindle rotation remains active until cancelled by:

- The opposite direction of rotation
- A SPINDLE STOP (M5 or M19)
- By END OF PROGRAM (M30) or CLEAR CONTROL.

PROGRAM STOP (M0/M1) OR TOOL CHANGE (M6/M66)

The machine tool interface determines whether the spindle rotation is suppressed or cancelled with a PROGRAM STOP or a TOOL CHANGE.

Suppressed means that the spindle starts rotating after resuming program execution.

Cancelled means that a spindle stop is executed and the direction of rotation has to be programmed again after a stop or tool change command.

Example

N20 G1 X100 Y100 S1000 M3

- N20: Start spindle rotating in a clockwise direction at 1000 rev/min before starting tool movement to programmed position.

10.3.4 M5 Spindle stop

Purpose

To stop spindle rotation. It depends on the interface of the machine tool if the coolant supply is switched off as well.

Format

N... {Programmed tool movement} M5

Associated functions

M3, M4, M19
M41, M42, M43, M44

Type of function

Modal

Notes and usage

ACTIVATION

The M5 function will become active when the current tool movement programmed in the same block has been executed.

The function remains active until a spindle rotation command is programmed.

Example

N35 G1 X50 Y50 F250 M5

N35: Execute tool movement and then switch off spindle rotation.

10.3.5 M6 Automatic tool change

Purpose

To interrupt program execution, retract the tool to a tool change position and perform a tool change, either automatically with a tool changer or manually.

Format

N... {T...} {T1=...} {T2=...} M6

Associated functions

M66, M67

Parameters

T	Tool identification number.
T1=	Activate/disable the cutting force monitor
T2=	Use the extra tool offsets

Type of function

Non-modal

Notes and usage

START TOOL CHANGE

The tool change is executed, before the tool movement programmed in the same block has been executed.

Note:

In the following description the most commonly used sequence for the tool change command is given. Refer to the machine tool builder's documentation to see:

- if a movement to a tool change position is executed by the interface,
- Which axes are involved.
- in which order the axes will move,
- if the spindle is stopped in an oriented position,
- if spindle speed and coolant supply are suppressed or cancelled.

MACHINE TOOL WITH AN AUTOMATIC TOOL CHANGER

The M6 function causes the following sequence to be performed:

- the tool first moves at rapid traverse rate to a tool change position.
- the old tool is then exchanged for a new tool and the new tools offsets made active.

RESUME PROGRAM EXECUTION AFTER THE AUTOMATIC TOOL CHANGE

The execution of the program continues automatically with the movement, if any, in the block with the tool change command.

MACHINE TOOL WITHOUT AN AUTOMATIC TOOL CHANGER

The M6 function causes the following sequence to be performed:

- the tool first moves at rapid traverse rate to a tool change position.
- the execution of the partprogram is halted, to allow the user to manually change the tool.

RESUME PROGRAM EXECUTION AFTER A MANUAL TOOL CHANGE

After the tool has been changed, pressing the START button restarts the partprogram. The movement, if any, in the block with the tool change command is executed.

SEARCHED TOOL (T)

If no T-word is programmed in a M6 block, the tool belonging to the last programmed tool number is loaded and its dimensions activated.

This situation occurs when the tool is searched during the execution of program blocks.

SPINDLE SPEED AND COOLANT SUPPLY

The machine tool interface determines whether the spindle rotation and coolant are suppressed or cancelled as well.

Suppressed means that the spindle starts rotating and the coolant supply is switched on after resuming the execution of the partprogram.

Cancelled means that a spindle stop is executed and the coolant supply switched off. These functions have to be programmed again after the tool change command.

TOOL CHANGE POSITION

It is advised to program all axes involved with the movement to a tool change position in the block with the tool change command or in the next block. In this way MANUAL BLOCK SEARCH and RESTART after program INTERRUPT is always executed in the same way.

INCREMENTAL PROGRAMMING AFTER A TOOL CHANGE

Increments with incremental programming are related to the last programmed position. A tool change position is not considered as a programmed position.

Example

N100: T12 M6

N100: Interrupt program execution to allow a new tool to be loaded. Tool no.12 offsets are made active.

10.3.6 M7/M8 Switch on coolant supply nr 2 / nr. 1**Purpose**

To switch on coolant supply
M7: Nr. 2 coolant supply (internal coolant supply)
M8: Nr. 1 coolant supply

Format

N... {Programmed tool movement} M7/M8

Associated functions

M9, M13, M14

Type of function

Modal

Notes and usage**ACTIVATION**

The coolant supply is switched on before the tool movement programmed in the same block, has been executed.

CANCELLATION

The function is active until cancelled by:

- COOLANT OFF (M9)
- END OF PROGRAM (M30) or CLEAR CONTROL.

SPINDLE STOP (M5)

It depends on the interface of the machine tool if the coolant supply is switched off with a SPINDLE STOP command.

PROGRAM STOP (M0/M1) OR TOOL CHANGE (M6/M66)

The machine tool interface determines whether the coolant supply is suppressed or cancelled with a PROGRAM STOP or a TOOL CHANGE.

Suppressed means that the coolant supply is switched on again after resuming program execution.

Cancelled means that the coolant supply is switched off and has to be programmed again after a stop or tool change command.

Example

N90: G1 X10 Y10 F200 M7

N90: Switch on No.2 coolant before executing programmed tool movement.

10.3.7 M9 Switch off coolant supply**Purpose**

To switch off simultaneously the two coolant supplies.

Format

N... {Programmed tool movement} M9

Associated functions

M7, M8, M13, M14

Type of function

Modal

Notes and usage**ACTIVATION**

The M9 function will become active when the current tool movement programmed in the same block has been executed.

The function remains active until one of the coolant functions (M7/M8, M13/M14) is activated again.

Example

N110: G1 X30 Y35 F150 M9

N110: Switch off coolant supplies after executing programmed tool movement.

10.3.8 M13/M14 Switch on nr. 1 coolant and rotate spindle CW/CCW**Purpose**

To switch on simultaneously the number 1 coolant supply and
M13 spindle rotation in clockwise direction
M14 spindle rotation in counter clockwise direction

Format

N... {Programmed tool movement} M13/M14

Associated functions

M3, M4, M5, M7, M8, M9
M41, M42, M43, M44

Type of function

Modal

Notes and usage**ACTIVATION**

The spindle starts rotating and the coolant supply is switched on before the tool movement programmed in the same block, has been executed.

CANCELLATION OF ROTATING SPINDLE

The spindle remains rotating until cancelled by:

- a SPINDLE STOP (M5 or M19)
- END OF PROGRAM (M30) or CLEAR CONTROL.

CANCELLATION OF COOLANT SUPPLY

The coolant supply remains switched on until cancelled by:

- COOLANT OFF (M9)
- END OF PROGRAM (M30) or CLEAR CONTROL.

SPINDLE STOP (M5)

It depends on the interface of the machine tool if the coolant supply is switched off with a SPINDLE STOP command.

PROGRAM STOP (M0/M1) OR TOOL CHANGE (M6/M66)

The machine tool interface determines whether the spindle rotation and coolant are suppressed or cancelled with a PROGRAM STOP or a TOOL CHANGE.

Suppressed means that the spindle starts rotating and the coolant supply is switched on after resuming the execution of the partprogram.

Cancelled means that a spindle stop is executed and the coolant supply switched off. These functions have to be programmed again after the stop or tool change command.

Note: A Machine Constant in the control must be set and the interface of the machine tool must be able to handle the functions M13 and M14. Refer to the machine tool builder's documentation to see if these functions can be programmed.

Example

N120 G1 X50 Y50 F100 S500 M13

Switch on no.1 coolant supply and rotate spindle clockwise at 500 rev/min before executing the programmed tool movement.

10.3.9 M19 Oriented spindle stop

Purpose

To stop the spindle in a fixed angular position, or, if an analogue spindle and a spindle transducer are used, in a programmed angular position.

Format

N... {D...} M19

Associated functions

M3, M4, M5, M13, M14

Parameter

D Spindle offset value

Type of function

Modal

Notes and usage

ANGULAR POSITION (D)

The angular position is measured from a fixed position, which is defined by IPLC.

DIRECTION OF ROTATION

Moving the spindle to the measured position always occurs in a fixed direction defined by a Machine Constant. (MC2412)

SIGN OF D-WORD

D+: the angle in the defined direction of rotation

D-: the angle (360° - D-value) in the defined direction.

Note: The D-word is only available if a Machine Constant is set. (MC2591)

ACTIVATION

The M19 function will become active when the current tool movement programmed in the same block has been executed.

The spindle remains in its position, until a spindle rotation command is programmed or a M19 with another position.

Example

N125: D30 M19

N125: Stop the spindle +30° from the fixed angular position.

10.3.10 M25 Measuring tool sizes**Purpose**

To measure tool sizes, using a measuring probe with cube-shaped probe tip.

The G50 function is used to change the stored tool sizes, if the recorded sizes are beyond the specified range of limit values.

Format

N... G45 [I / J / K] X1=... M25

Associated functions

-

Parameter

I	+/-	2.3	Measuring direction for X axis
J	+/-	2.3	Measuring direction for Y axis
K	+/-	2.3	Measuring direction for Z axis
L	+/-	2.3	Measuring direction for B axis
X1=	+/-	6.3 5.4	Pre-measurement distance (mm inch)

Type of function

Non-modal

Notes and usage**G45 MEASURING CYCLE**

- A measuring probe with cube-shaped probe tip is used for tool measurement. The probe is mounted in a fixed position.
- The measuring position is loaded to the CNC memory.

MACHINE CONSTANT MEMORY

The following is stored in this memory:

- the coordinates of the fixed position of the measuring probe
- The sizes of the cube-shaped measuring probe tip.

Example

N90 G45 -I X1=5 M25

N90: Measure the tool in negative direction of the X-axis. The pre-measurement distance is 5 mm.

10.3.11 M26 Calibration the measuring probe

Purpose

The measuring probe radius is established by probing a calibration ring (ring gauge whose diameter is exactly known).

Format

Using the outer face of the ring gauge:

N... G46 [I+1,J+1 / J+1,K+1 / I+1,K+1] T... X1=... F... M26

Using the inner face of the ring gauge:

N... G46 [I-1,J-1 / J-1,K-1 / I-1,K-1] T... X1=... F... M26

I and J - gauge in XY plane; J and K - gauge in YZ plane; I and K - gauge in XZ plane

Parameters

I	+/-	1	Measuring direction for X axis (J or K should also be indicated)
J	+/-	1	Measuring direction for Y axis (I or K should also be indicated)
K	+/-	1	Measuring direction for Z axis (I or J should also be indicated)
F	+/-	4.3 3.4	Feed rate (mm/min inch/min)
T		7.2	Tool number
X1=	+/-	6.3 5.4	Pre-measurement distance (mm inch)

Type of function

Non-modal

Notes and usage

MEASURING CYCLE

The G46/M26 measurement cycle is similar to the G46 measurement cycle. The difference between the measured centre point and the centre point stored in the machine constant memory is calculated. The F50 function is able to use this value for zero point shifts.

MACHINE CONSTANT MEMORY -(MC242 / MC292 etc.).

The following is stored in this memory:

- the coordinates of the fixed position of the ring gauge
- the ring gauge diameter.

If X1= has not been programmed in the M26 block, a fixed preset value (machine constant) is used.

Example

N190 G46 I-1 K-1 T15 F50 M26:

Calibrate the probe by moving it to the outer face of the ring gauge in the XZ plane. The probe radius is stored in tool memory location T15. A fixed preset value (machine constant) is used for X1=.

10.3.12 M24/M27/M28 Switch on/off a measuring probe**Purpose**

Before measurements can be performed with a remote signalling probe, e.g. an infrared probe, or a hard wired probe, the probe must be switched on and after using the probe, it must be switched off.

M24:switch on the measuring probe (Measuring probe stands on the table)

M27:switch on the measuring probe (Measuring probe is mounted in the spindle)

M28:switch off the measuring probe

Format

Switch on a probe

N... M24

or

N... M27

Switch off a probe

N... M28

Associated functions

G145, Measuring tool G600-G609

Type of function

Modal

Notes and usage**ACTIVATION**

The function M27 is executed before the movements programmed in the same block, are executed and M28 after the movements in the block.

CANCELLATION

The measuring probe is switched off with the function M28, the function M30 (end of program) or at CLEAR CONTROL.

10.3.13 M30 End of partprogram

Purpose

To terminate the execution of the partprogram and jump back to the begin of the program.

Format

N... M30

Type of function

Non-modal

Notes and usage

SPINDLE ROTATION AND COOLANT SUPPLY

When a M30 block is executed, the spindle is stopped and coolant supply switched off by the control.

DEFAULT SETTINGS

From G-functions belonging to one group, the default function of that group, if any, is automatically activated when the M30 function is executed.

Other functions with a default setting are reset too.

Example

```
N9001  
N1 ...  
:  
N... M30
```

Explanation

N9001: Partprogram identification block.

N1 to N...: Partprogram instructions.

N... End of partprogram and jump back to begin of program

10.3.14 M41/M42/M43/M44 Select spindle speed range**Purpose**

To select the gear range for the required spindle speed. In general the gear range is automatically selected by the control. In exceptional cases it may be necessary to change the speed without changing the gear range. This is done using the functions M41 - M44.

M41:	first speed range	(MC4425)
M42:	second speed range	(MC4445)
M43:	third speed range	(MC4465)
M44:	fourth speed range	(MC4485)

Format

N... S... M41/M42/M43/M44

Associated functions

M3, M4, M5, M13, M14, M19

Type of function

Modal

Notes and usage**SPEED RANGE SELECTION**

The speed range can be selected:

- Automatically by the CNC; the corresponding M-function is produced automatically by the CNC.
- By programming the corresponding M-function; useful when overlapping speed ranges are used.

SPEED RANGE LIMITS

The limits of the spindle speed ranges are stored in the MC-memory of the CNC.

TYPE OF SPEED RANGES

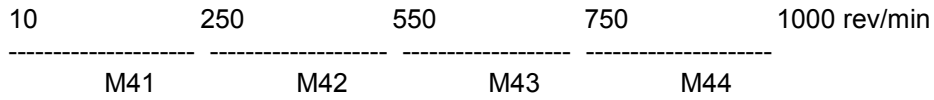
The spindle speeds can either be in separate speed ranges or in ranges, which overlap each other.

If the M-function for range selection is not programmed and a programmed spindle speed occurs in two ranges, the highest range is automatically selected.

Example

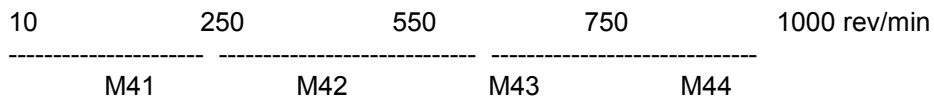
Example of speed ranges which do not overlap.

M41: 10 - 250 rev/min;
 M42: 250 - 550 rev/min;
 M43: 550 - 750 rev/min;
 M44: 750 - 1000 rev/min.



Example of speed ranges which overlap.

M41: 10 - 250 rev/min;
 M42: 200 - 550 rev/min;
 M43: 500 - 750 rev/min;
 M44: 700 - 1000 rev/min.



Programming example

N10: S50 M41

N10: Assumed are the speed ranges given above. A spindle speed of 50 rev/min is required. Therefore M41 is programmed because spindle range 1 has to be used. Automatic range selection is not being used.

10.3.15 M66 Manuel tool change

Purpose

To interrupt program execution without a retract to a tool change position, to allow a manual tool change to be performed.

Format

N... T... {T1=...} {T2=...} M66

Associated functions

M6, M67

Parameter

T	Tool identification number.
T1=	Activate/disable the cutting force monitor
T2=	Use the extra tool offsets

Type of function

Non-modal

Notes and usage

APPLICATION OF M66

The function M66 is used with a tool:

- which is not in the tool magazine,
- for e.g. a back boring operation

START TOOL CHANGE

The tool change is executed, before the tool movement programmed in the same block has been executed.

MACHINE TOOL WITH AUTOMATIC TOOL CHANGER

The M66 function is used, when a tool is required which is not in the tool magazine. There is no automatic retraction to the tool change position and no execution of the tool change sequence.

Before performing manual tool change it might be necessary to unload the spindle (by programming T0 M6) and put the tool back in the tool magazine.

It might also be necessary to program a retract to a position where the tool can be loaded.

MACHINE TOOL WITHOUT AUTOMATIC TOOL CHANGER

The M66 function causes a halt in the program execution without a retraction to the tool change position, to allow the tool to be manually changed.

RESUME PROGRAM EXECUTION

After the tool change the program is restarted by pressing the START button. The movement, if any, in the block with the tool change command is executed.

SPINDLE SPEED AND COOLANT SUPPLY

The machine tool interface determines whether the spindle rotation and coolant are suppressed or cancelled as well.

Suppressed means that the spindle starts rotating and the coolant supply is switched on after resuming the execution of the partprogram.

Cancelled means that a spindle stop is executed and the coolant supply switched off. These functions have to be programmed again after the tool change command.

Example

N200 T24 M66

N200: Interrupt program execution and change the tool manually. The tool dimensions of T24 become active.

10.3.16 M67 Change tool compensation values**Purpose**

To activate tool compensation values without a change of the physical tool being performed.

Format

N... T... {T2=...} M67

Associated functions

M6, M66

Parameters

T Tool identification number.
T2= Use the extra tool offsets

Type of function

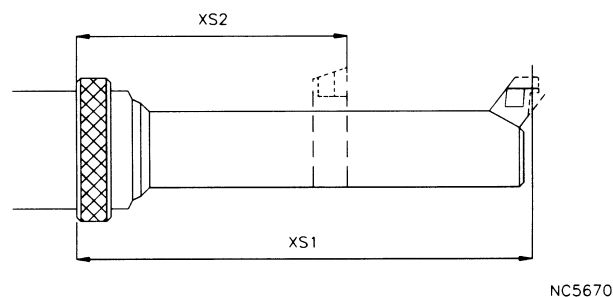
Non-modal

Notes and usage**TOOLS WITH MORE THAN ONE CUTTING EDGE**

When a tool with more than one cutting edge is used, e.g. a boring bar, each cutting edge has its specific length and radius which are stored in the tool memory as offsets and extra tool offsets for the same tool.

ACTIVATE TOOL DIMENSIONS

The new tool dimensions are activated, before the tool movement programmed in the same block has been executed.

Example

NC5670

The above boring bar, identified as tool T12, has two cutting edges. Edge 1 with tool length XS1 is stored in the tool memory with L=XS1 and edge 2 with tool length XS2 with L1 =XS2.

N100 T12 M6
N110 - N140
N150 T12 T2=1 M67

Explanation

N100: Load the boring bar; the offset values from T12 are used.
N110 - N140: Use cutting edge defined with T12
N150: A change of the offset values from XS1 to XS2. The boring bar itself is not changed.

10.4 S-function

Purpose

To set the speed in revolutions per minute (rev/min) of the main spindle (S) or second spindle (S1=) of the machine tool. The speed that can be used depends on a number of factors, such as material, operation, tool etc.

Format

N... {S....} {S1=...}

Associated functions

M3, M4, M5, M13, M14, M19
M41, M42, M43, M44
M1=3, M1=4, M1=5

Type of function

Modal

Notes and usage

MAXIMUM VALUE

A maximum speed of 50000 rev/min. can be programmed with the S- WORD.

Refer also to the documentation of the machine tool builder to see which spindle speeds can be used on the machine tool.

The maximum spindle speed of the second spindle is set in MC2691

DIRECTION OF SPINDLE ROTATION

Refer to the description of M3/M4 for programming the direction of spindle rotation.

SPINDLE SPEED RANGES

Refer to the description of M41/M42/M43/M44 for selecting a spindle speed range.

TECHNOLOGY TABLES

If the partprogram is entered or updated via the control, it is possible to retrieve the spindle speed from the technology tables stored in the control.

The material, type of operation and tool has to be entered too to make the selection in these tables.

Refer to the User manual for details about using the technology tables.

Example

N10: S1000

N10: Set the spindle speed 1000 rev/min.

10.5 T-function tool number and tool memory

- | | |
|----|--------------------------------|
| a. | Description of the T-function |
| b. | Description of the tool memory |

10.5.1 T-function

Purpose

To enter the tool identification number in the CNC Tool Memory. To indicate the tool to be loaded and call its offset values during the execution of the program. To initiate a search for another tool in the tool magazine.

Format

Automatic tool change
N... {T...} {T2=1/2} {T1=...} M6

Manual tool change
N... T... {T2=1/2} {T1=...} M66

Change offset values
N... T... {T2=1/2} M67

Tool change with sufficient tool life
N... T... T3=.. M6/M66

Associated functions

M6, M66, M67

Parameter

T	Tool identification number
T1=	Activate/disable the cutting force monitor
T2=	Use the extra tool offsets
T3=	Search for a tool with sufficient tool life

Type of function

Non-modal

Notes and usage

THE TOOL MEMORY

A special memory is available in the control to store the tool dimensions and other tool related parameters. Refer to the appendix TOOL MEMORY for a description of this memory.

MAXIMUM NUMBER OF TOOLS

A maximum of 255 sets of tool dimensions can be stored in the tool memory of the control. The actual number is stored as a machine constant.

TOOL IDENTIFICATION NUMBER (T)

The tool number in the tool memory is used to identify the tool. It is entered with the address T and a value with eight digits before and two digits behind the decimal point. The eight digits are reserved for the tool identification number. This number is also used in the partprogram and programmed with the T-words.

The two digits behind the decimal point specify a spare tool related to the tool.

If necessary these two digits can be used in the partprogram as well.

EXTRA TOOL OFFSETS (T2=)

A tool can have extra offsets. So the tool length, the tool radius and the corner radius of a mill can have three values: L, L1=, L2=, R, R1=, R2= and C, C1=, C2= respectively.

Which offsets are used when the tool is loaded, is indicated with the word T2= in the partprogram.

T2= not programmed: the offsets stated by L, R and C are used.

T2=1: the offsets stated by L1=, R1= and C1= are used. T2=2: the offsets stated by L2=, R2= and C2= are used.

E.g. a program block containing the words T1234 T2=2 M6 results in tool number 1234 being loaded together with its second offsets.

TOOL CHANGE WITH SUFFICIENT TOOL LIFE (T3=)

The T3= word in the partprogram indicates which replacement tool with sufficient residual tool life is used for a tool change.

A tool may have several replacement tools with different residual tool lives. (Refer to TOOL LIFE MONITORING in Appendix "TOOL MEMORY"). When T3= is programmed in a search block or during a tool change, a tool with sufficient residual tool life is searched.

Error P117 is displayed, if no tool with sufficient residual tool life is found.

Example:

A program block containing the words T1234 T3=1.1 M6 causes tool No. 1234 to be changed. The residual tool life is at least 1.1 minutes.

Note: If the life of a replacement tool expires during the machining operation, the program run is interrupted by error message P118. The program run will be resumed when the error message has been deleted and the Start key pressed. This error message is only activated if no other tools are changed during the operating sequence of Tool selection, Tool life run, Replacement tool selection and Replacement tool life run. To avoid this fault condition, the programmer may program an empty block preceding the tool change block, or he can avoid the situation described.

Note: If replacement tools outside the tool magazine range have been stored in the tool memory (location > MC28), error message P117 may be activated if a replacement tool is selected during the program run. To avoid this, the replacement tools in the tool magazine range should be stored in the tool memory. This applies, at the very least, to the replacement tools that may be used during the program run. If several replacement tools are envisaged for a particular tool in the tool memory, the tools with the lowest number of allocated replacement tools should be stored in the tool magazine range.

TOOL CHANGE COMMANDS (**M6, M66, M67**)

A physical tool change is commanded by one of the functions M6 (automatic tool change) or M66 (manual tool change) and a change of tool offsets by M67. Refer to these functions for details how to use them.

UNLOADING THE SPINDLE

With T0 M6 the spindle is unloaded and the tool put back at the position it originally left.

Unloading the spindle is necessary:

- before a manual tool change
- with oversized tools.

TOOL SEARCH

During the execution of a program, it is possible to search for the next tool in the magazine. So the tool is ready, when it should be loaded.

If the T-word is programmed without a tool change command, the search for the next tool is activated, provided that the interface of the machine tool allows a tool search.

SPARE TOOLS

A spare tool can replace the tool after its working life has ended or the lowest power level of the tool in the cutting force monitor is exceeded.

The spare tool is a two-digit number placed behind the decimal point of the tool identification number.

SELECTING A SPARE TOOL

When a program block containing the words T1 M6 is executed, the CNC searches in the tool memory for the tool T1.xx.

If the tool with the lowest spare number has not been disabled, it is selected. If it cannot be used, another spare tool is selected which has a higher spare number.

If tool identification and the spare tool number are programmed, e.g. T1.05, spare tool 05 of tool 1 is used.

TOOL LIFE MONITORING

A working tool life is assigned to a tool. Every time the tool is used, the tool life is reduced with the cutting time. When the tool life has expired, a warning message is displayed, so that the tool can be replaced.

Refer to the appendix TOOL MEMORY for a description of the tool life monitoring and the actions taken when the tool life is exceeded.

TOOL BREAKAGE MONITOR

With an external device mounted on the machine tool, the tool length is measured when the tool is loaded into the spindle and again when it is put back into the magazine. If the difference between the two measurements is greater than a tolerance value, an error message is displayed and the tool disabled.

Refer to the appendix TOOL MEMORY for a description of the tool breakage monitoring and the actions taken when the tolerance value is exceeded.

CUTTING FORCE MONITORING

With an external device mounted on the machine tool, the cutting force being applied to a tool can be monitored by constantly measuring the power consumption of the spindle drive. When a power overload condition is detected, appropriate actions will be taken to prevent the workpiece or tool from being damaged.

ACTIVATING THE CUTTING FORCE MONITOR (T1=)

The cutting force monitor is activated with the word T1= and a value from 1 to 99 indicating the group of power levels to be used. The T1= word is programmed in a block containing one of the tool change functions (M6 or M66).

DISABLING THE CUTTING FORCE MONITOR

The cutting force monitor is disabled by programming T1=0 or by not programming the T1= word.

Notes

1. Cutting force monitoring is usually used with heavy cutting, generally with tools of >10 mm diameter.
2. Refer to the appendix TOOL MEMORY for a description of the power levels with a cutting force monitor and the actions taken when a power level is exceeded.

10.5.2 Tool memory

Purpose

The tool memory of the control can be used to store tool dimensions and other tool related parameters. This memory can be used in a FMS environment, but also outside such an environment.

In this section a general description of the parameters in the tool memory is given.

Refer also to the machine tool builder's documentation to see which monitoring devices are activated on your machine tool, Refer also to the user manual for entering the tool data into the memory.

Addresses in tool memory:

P, T, T2=1/2, T1=, T3=, L, L1=, L2=, R, R1=, R2=, C, C1=, C2=, L4=, R4=, L5=, R5=, L6=, R6=, G, Q3=, Q4=, I2=, A1=, E, S, M, M1=, M2=, B, B1=, Q5=, O

P	Place of the tool in the magazine
T	Tool number
L	Length
R	Radius
C	Corner radius
L4=	Length oversize
R4=	Radius oversize
G	Graphics
Q3=	Type
Q4=	Number of cutting edges
I2=	Cutting direction
A1=	Approach angle
S	Size
E	Status
M	Initial tool life
M1=	Actual tool life
M2=	Tool life monitoring
B	Breakage tolerance
B1=	Breakage monitoring
L1=	First extra length
R1=	First extra radius
C1=	First extra corner radius
L2=	Second extra length
R2=	Second extra radius
C2=	Second extra corner radius
L5=	Wear tolerance length
R5=	Wear tolerance radius
L6=	Offset length
R6=	Offset radius
Q5=	Breakage monitoring cycle (0-9999)
O	Tool orientation (only turning)

Description of the addresses

RANDOM ACCESS TOOL MAGAZINE

When a tool magazine can be filled at random, a table containing per tool its place in the magazine and the corresponding tool identification number should be stored in the tool memory of the control before the first run of the program.

At a tool change (M6) the programmed tool is picked up from the magazine and the used tool put back at the empty place of the loaded tool. The table of tool places is automatically updated by the control.

PLACE OF TOOL IN MAGAZINE (P)

The three digits P-word in the tool memory are used for indicating the place of the tool in the magazine, where P1 corresponds to place 1, P2 to place 2, etc.

The actual number of places in the magazine is stored as a Machine Constant.

TOOL IDENTIFICATION NUMBER (T)

The tool number in the tool memory is used to identify the tool. It is entered with the address T and a value with eight digits before and two digits behind the decimal point. The eight digits are reserved for the tool identification number.

The two digits behind the decimal point specify a spare tool related to the tool.

SPARE TOOLS

A spare tool can replace the tool after its working life has ended or the lowest power level for this tool in the cutting force monitor, if available, is exceeded.

The spare tool is a two-digit number placed behind the decimal point of the tool identification number. Therefore, a maximum of 99 spare tools can be assigned to the same tool.

ACTIVATING THE CUTTING FORCE MONITOR (T1=)

The cutting force monitor is activated with the word T1= and a value from 1 to 99 indicating the group of power levels to be used. The T1= word is programmed in a block containing one of the tool change functions (M6/M66).

EXTRA TOOL OFFSETS (T2=)

A tool can have extra offsets. So the tool length, the tool radius and the corner radius of a mill can have three values: L, L1 =, L2=, R, R1=, R2= and C, C1=, C2= respectively.

Which offsets are used when the tool is loaded, is indicated with the word T2= in the partprogram.

T2= not programmed: the offsets stated by L, R and C are used.

T2=1: the offsets stated by L1=, R1= and C1= are used. T2=2: the offsets stated by L2=, R2= and C2= are used.

E.g. a program block containing the words T1234 T2=2 M6 results in tool number 1234 being loaded together with its second offsets.

TOOL DIMENSIONS (**L**, **L1=**, **L2=**, **R**, **R1=**, **R2=**, **C**, **C1=**, **C2=**)

A tool can have a length (L-word) and two extra length values (L1 = and L2=), a radius (R-word) and two extra radius values (R1= and R2=) and a corner radius (C-word) and two extra values (C1= and C2=).

Refer to the T-word EXTRA TOOL OFFSETS (T2=) for activating the extra tool offsets.

LENGTH- AND RADIUS OVERSIZE (**L4=**, **R4=**)

These parameters are to define the extra oversize of the tool form. The real tool length is the length (L) plus the length oversize (L4=). The real radius of the tool is the radius (R) plus the radius oversize (R4=). The length- and radius compensation works with these length- and radius-values. When no data is entered, the values are zero.

Note: The length value is used with the length compensation, the radius value with radius compensation (G41 to G44 or G141), the corner radius is used with 3D radius correction (G141).

OVERSIZED TOOLS (**S**)

With the S-word in the tool memory is indicated if a tool occupies one place (S0) or is oversized (S1). In the latter case the tool occupies three places in the magazine, one place in which the tool is stored and two empty places at the left and right of the tool.

After using such a tool it should be put back at the same place in the magazine.

GRAPHIC PARAMETER (**G**)

This parameter is used to define the tool shape. The available shapes are displayed when entering tool data into the tool memory.

The tool shape is used with the graphical simulation of a partprogram.

TOOL TYPE (**Q3=**)

The tool type parameter in the tool memory (Q3=) has to be the same as the Q3= parameter in the technology table.

If the material, type of operation and tool number is entered, when asking for a technology proposal, this parameter is automatically picked up from the tool memory.

NUMBER OF CUTTING EDGES (**Q4=**)

This parameter indicates the number of cutting edges of a mill. If a technology proposal is asked for during entering a program via the control, this parameter is picked up from the tool memory, provided that the tool number and Q3= parameter are already entered.

CUTTING DIRECTION (**I2=**)

This parameter indicates the cutting direction (3=M3, 4=M4). When no data is entered, M3 is active

APPROACH ANGLE (**A1=**)

This parameter indicates the maximum angle for entering in the material. When this value is 90, the mill goes down perpendicular into the material.

ENABLING/DISABLING A TOOL (E)

The E-word in the tool memory indicates if the tool can be used or not.

E-1: tool is disabled, cannot be used. This parameter is set by the control if the tool life is ended or the lowest power level of the cutting force monitor is exceeded.

E0: tool can be used, but is not measured;

E1: tool can be used and is measured.

WEAR TOLERANCE (LENGTH AND RADIUS) FOR MEASUREMENT CYCLES (L5=, R5=)

These parameters give the maximum limits for the wear tolerances by tool measurement cycles. When by measurement the wear greater is then these tolerances an error message is given.

TOOL MEASUREMENT OFFSETS FOR MEASUREMENT CYCLES (L6=, R6=)

These parameters indicate the positions, where the length or the radius must be measured. L6= contains the distance between the tool length (L) and the measure position.

TOOL LIFE MONITORING (M, M1=, M2=)

With the word M2= in the tool memory is indicated that the tool life of a specific tool should be monitored by the CNC.

M2=0: no tool life monitoring for the specific tool

M2=1: tool life should be monitored

The word M in the tool memory is available to assign a working life in minutes to a tool. The stored tool life ranges from 1 to 99999 minutes.

Every time the tool is used, the working tool life in the memory is reduced with the cutting time. With the word M1= in the tool memory the remaining tool life is displayed.

When the tool life has expired a warning message is displayed, so that the tool can be replaced and the E-word in the tool memory set to -1 to indicate that the tool is disabled.

When the same tool is used again, program execution is either interrupted or a spare tool loaded (depending on the machine tool configuration).

A Machine Constant must be set to indicate that tool life monitoring for the tools is required.

TOOL CHANGE WITH SUFFICIENT TOOL LIFE TIME (T3=)

With the address T3= in the program will be indicated, which tool with sufficient rest tool lifetime must be taken by the tool change. A tool can have more spare tools with difference rest tool lifetime. With the programming of T3= in a search block or by a tool change, will be chosen the tool which has over a sufficient tool lifetime. When no tool with sufficient rest tool lifetime will be found, an error message is given (P117).

For example:

A program block with the addresses T1234 T3=1.1 M6 start the tool change of tool 1234.??, where by the minimum rest tool lifetime must be 1.1 minutes.

TOOL BREAKAGE MONITOR (B, B1=)

With an external device mounted on the machine tool, the tool length is measured when the tool is loaded into the spindle and again when it is put back into the magazine. If the difference between the two measurements is greater than a tolerance value, an error message is displayed and the tool disabled.

The tolerance value is entered into the tool memory with the B-word.

With the word B1= in the tool memory is indicated that the tool breakage of a specific tool should be monitored by the CNC.

B1=0: no tool breakage monitoring for the specific tool

B1=1: tool breakage should be monitored

A Machine Constant must be set to indicate that tool breakage monitoring for the tools is required.

CUTTING FORCE MONITOR

With an external device mounted on the machine tool, the cutting force being applied to a tool can be monitored by constantly measuring the power consumption of the spindle drive. When a power overload condition is detected, appropriate actions will be taken to prevent the workpiece or tool from being damaged.

Note

1. Cutting force monitoring is usually used with heavy cutting, generally with tools of >10 mm diameter.
2. Refer to the description of the T-word, Notes and Usage CUTTING FORCE MONITOR for activating and disabling the cutting force monitor in a program.
3. A Machine Constant must be set to indicate that cutting force monitoring for the tools is required.

CUTTING FORCE LEVELS

A group of three cutting force levels can be chosen for a tool. These levels are determined by such factors as the type of tool, the workpiece material and the type of machining operation to be performed. Up to 99 groups of power levels can be stored in the monitor's memory.

The monitoring system generates output signals to indicate which power level is exceeded by the cutting force.

LOWEST POWER LEVEL EXCEEDED

When the cutting force exceeds the lowest power level, this indicates that the tool is worn out. The CNC generates a warning message but continues using the tool until the next tool change or another power level is exceeded. The E-word in the tool memory is set -1 to indicate that the tool is disabled.

If the tool is used again an error message is generated or, if available, an assigned spare tool loaded.

MEDIUM POWER LEVEL EXCEEDED

When the cutting force exceeds the medium power level, the CNC responds to this message with an immediate feed/speed hold and an error message. The user has to intervene.

HIGHEST POWER LEVEL EXCEEDED

When the cutting force exceeds the highest power level, this indicates that the maximum permitted cutting force has been exceeded and an emergency stop is made. The user has to intervene.

BREAKAGE MONITORING CYCLE (Q5=)

By hand this address it is possible to make decision in the IPLC. For example: which breakage monitoring cycle must be used. See the machine builder handbook.

TOOL ORIENTATION (O) (Only turning)

For turning tools it is possible to tell with the tool orientation (O) in which direction the tooltip is orientated..

Refer to paragraph 12.7 "Tools for turning mode" on page 559.

11. E-Parameters and arithmetic functions

11.1 E-Parameter

Purpose

E-parameters are useful because they allow a more flexible use of programs: one program can be used for producing different workpieces by changing the parameter values stored in the CNC's Parameter Memory.

With the aid of macros and E-parameters a problem can be solved in general terms, e.g. measuring a round hole in three or four points. At execution time the parameters receive their actual values and so the macro is adapted to the specific requirements of the program, e.g. measuring a hole in three points.

Format

Parameter definition
N... E..=[Value or arithmetical expression]

Parameter allocation
N... [Address]= {+/-} E...

Parameter allocation and calculation
N... [Address]=[arithmetical expression]

Associated functions

G14, G22, G29

Type of function

Modal

Notes and Usage

CANCELLATION

Parameter values are modal unless changed by:

- Assignment of new values in a partprogram,
- Input via the user's panel in the parameter memory,
- Input from a data carrier in the parameter memory.

Parameter values are not removed by Softkey <CANCEL PROGRAM> or Softkey <CLEAR CONTROL> or M30.

NUMBER OF PARAMETERS

A maximum of 400 parameter values can be stored: a Machine Constant can be used for reducing this number.

ADDRESS

Any of the addresses available, except the address 'N'.

PARAMETER NUMBER (E)

This number specifies where the numerical value is stored in the CNC's Parameter Memory.

USE OF A PARAMETER IN MORE THAN ONE PROGRAM

Different programs can use a parameter. If a new program uses a parameter, which already has been assigned a value by a previous program, a new value must be assigned to the parameter otherwise the old value will be used again.

When a parameter is programmed but no value for that parameter is present in the Parameter Memory, an error message is produced.

STANDARD TYPES OF PARAMETERS

Parameters can be used in every CNC PILOT system. If the option EXTENDED ARITHMETICAL OPERATIONS is not available, the type of the parameter can be:

- Integer, no decimal point: E1=20
- Fixed point value: E1=200.105

INPUT ACCURACY WITH THE STANDARD TYPES

The input accuracy of the standard parameter types is:

- Integer: a 15 digit number
- Fixed point: at least 6 decimals behind the decimal point
Maximum 15 decimals behind the decimal point

DISPLAY OF PARAMETER VALUES

The parameters stored in the Parameter Memory can be displayed on the screen of the control.

The displayed values are rounded values with a restricted number of decimals. They are displayed either as a fixed-point value or as a value in the so-called scientific notation, thus with an exponent.

Though the stored value has the same or a greater accuracy as the displayed one, the stored value might be different.

E.g. stored in the memory: 99.99999999
Displayed: 100

STANDARD ARITHMETICAL OPERATIONS

The four arithmetical operations of addition (+), subtraction (-), multiplication (*) and division (:) are available in every CNC system.

E1=E2: set E1 value equal to E2 value.
E1=E2+E3: add the E3 value to the E2 value and store the result in E1.
E1=E2-E3: subtract the E3 value from the E2 value and store the result in E1
E1=E2*E3: multiply the E2 value by the E3 value and store the result in E1
E1=E2:E3: divide the E2 value by the E3 value and store the result in E1.

RESTRICTIONS

1. Arithmetical expressions must not contain spaces between characters.
E.g. E1 = E2 is not allowed. This should be E1=E2
2. Arithmetical operators must be between arithmetical values e.g.
E1=E2 E3 is not allowed. Must be E1=E2*E3.
3. Consecutive arithmetical operators are also not allowed e.g.
E1=E2*:E3, except in the case of E1=E2*-E3.

Only one arithmetical operation is allowed in an expression.

11.2 Arithmetical functions

Format

Arithmetical operations

Exponentiation	$E1=E2^E3$
Square root	$E1=\text{sqrt}(E2)$

Note: In all the cases, it is permitted to replace the E-Parameter between the parentheses by an arithmetical expression (up to four levels) e.g. $E1=\text{sqrt}(E2^2+E3^4)$.

Absolute value	$E1=\text{abs}(E2)$
Integer conversion	$E1=\text{int}(E2)$
Value of Pi(=3.141592..)	pi

Extended from V420:

Whole number conversion with large value	$E1=\text{ciel}(E2)$
Whole number conversion with small value	$E1=\text{floor}(E2)$
Rounding	$E1=\text{round}(E2,n)$ (n is decimals)
Remainder of division	$E1=\text{mod}(E2,E3)$
Sign	$E1=\text{sign}(E2)$
Maximum	$E1=\text{max}(E2,E3)$
Minimum	$E1=\text{min}(E2,E3)$

Trigonometrically and inverse trigonometrically functions

Sine	$E1=\sin(E2)$
Cosine	$E1=\cos(E2)$
Tangent	$E1=\tan(E2)$
Arc sine	$E1=\text{asin}(E2)$
Arc cosine	$E1=\text{acos}(E2)$
Arc tangent	$E1=\text{atan}(E2)$

Extended from V420:

Arc sine	$E1=\text{asin}(E2,E3)$
Arc cosine	$E1=\text{acos}(E2,E3)$
Arc tangent	$E1=\text{atan}(E2,E3)$

Relational expressions

equal	=	$E1=E2=E3$
not equal	<>	$E1=E2<>E3$ true $E1=1$
greater	>	$E1=E2>E3$
greater or equal	>=	$E1=E2>=E3$ not true $E1=0$
less	<	$E1=E2<E3$
less or equal	<=	$E1=E2<=E3$

Parentheses (...(...(...(...)...)...): maximum of four levels

Notes:

1. If the relational expression is true $E1=1$, if not true $E1=0$. This can be used in the function for conditional jump (G29)
2. On a data carrier the mentioned functions should be in lower case characters.
3. In the format description, the parameters E2 and E3 represent any parameter or expression.
4. Functions and arithmetical expressions can also be used without parameters, e.g. $X=(10+12*\sin(23))$.
5. The E-parameter containing the result of a calculation of a mathematical function, which has the required accuracy, but different decimals can be stored.

E.g. E1=99.9999999 and E1=100.0000001 are two values with the same accuracy, but different decimals.

Problems may arise when using the function "int" or a relational expression in which all digits are compared.

BLOCK LENGTH

The size of an expression is restricted to 40 characters.

A program block can contain a maximum of 255 characters. This restricts the number of expressions that can be in a program block.

CONVERTING CALCULATED VALUES TO PROGRAM WORDS

Parameter (or calculated) values are automatically 'rounded' and converted by the CNC to the fixed number of decimals belonging to the program word.

E.g. programming E1=101.74e-3 and X=E1 causes the value to be rounded, so that X0.102 is obtained. The value is reduced to three digits after the decimal point.

EXTENDED PARAMETER FORMATS

A parameter value may be input in one of three formats: integer, fixed point and floating point.

- | | |
|--------------------------------------|------------|
| 1. Integer value (no decimal point): | E1=20 |
| 2. Fixed point value: | E1=200.105 |
| 3. Floating point value: | E1=1.965e5 |

A floating-point value is comprised of a fixed-point number (the mantissa) which is multiplied by an exponential value e.g. 1.965e5 means $1.965 \times (10^5)$, which is equal to 196500.

INPUT ACCURACY WITH THE EXTENDED FORMAT

The input accuracy of the parameter types is:

- Integer: a 15-digit number
- Fixed point: at least 6 decimals behind the decimal point, maximum 15 decimals behind the decimal point
- Floating point: The mantissa is programmed as a fixed-point value; the exponent is an integer between -99 to +99.

EXPONENTIATION (RAISING TO A POWER)

E1=E2^2 or E1=E2^E3 (with E3=2)

Both of the above operations result in the E1 parameter being made equal to the square of the E2 value.

Exponentiation operations are performed in a fixed sequence. The exponentiation operation is performed first and then the effect of the sign is included.

For example, the equation of E1=-3^2 is evaluated by first performing exponentiation (3^2) and then including the effect of the sign resulting in a negative value (-9).

If a negative number has to be raised to a power, the number should be enclosed by parentheses e.g. E1=(-3)^E3. Another method is to assign the negative number to a parameter and then perform the exponentiation operation on the parameter e.g. E2=-3 and then E1=E2^2.

Not allowed exponentiation calculations are:

1. 0^0 ;
2. E2^E3, when E2<0 and E3 has a real value.

RECIPROCAL

The reciprocal of E2 can be calculated by $E1=1:E2$ or $E1=E2^{-1}$

QUADRATE

The quadrate of E2 can be calculated by $E1=E2 \cdot E2$ or $E1=E2^2$

SQUARE ROOTS

The square root of E2 can be calculated by:

$E1=\text{sqrt}(E2)$ or $E1=E2^{.5}$

$E1=\text{sqrt}(\dots)$: between parentheses an arithmetical expression is allowed e.g.

$E1=\text{sqrt}(E2^2+E3^4)$.

Parameter E2 must be positive or zero when a square root (sqrt) calculation is performed.

ABSOLUTE VALUES

When the absolute function is used a negative value becomes positive. Positive values remain unchanged.

$E1=\text{abs}(E2)$

WHOLE NUMBERS

When using the integer function, the numerical value is rounded, i.e. all figures after the decimal point are ignored.

$E1=\text{int}(E2)$

Example: $E2=8.9$ results in 8, $E2=-8.9$ results in -8

Note: The integer function is changed with the floor function in V420 and higher

WHOLE NUMBER WITH LARGEST VALUE

When using the integer function with the largest value, the numerical value is rounded according to the smallest argument.

$E1=\text{ciel}(E2)$

Example: $E2=8.9$ results in 8, $E2=-8.9$ results in -9

WHOLE NUMBERS WITH SMALLEST VALUE

When using the integer function with the smallest value, the numerical value is rounded according to the smallest argument.

$E1=\text{floor}(E2)$

Example: $E2=8.9$ results in 9, $E2=-8.9$ results in -8

ROUNDING

When the rounding function is used, the numerical value is rounded according to the number of decimal places.

$E1=\text{round}(E2,n)$ (n is number of decimal places)

Remark: If the number of decimal places is not entered, zero is assumed.

Example: $n=1$ and $E2=8.94$ results in 8.9, $n=1$ and $E2=-8.94$ results in -8.9
 $n=1$ and $E2=8.96$ results in 9.0, $n=1$ and $E2=-8.96$ results in -9.0

REMAINDER OF DIVIDING

When the remainder function is used, the remainder is returned by the argument.

$E1=\text{mod}(E2,E3)$

Remarks:

- $E1=E2-\text{int}(E2:E3) \cdot E3$
- If E3 is 0, E2 is returned.
- If E3 is not entered, 1 is assumed.

The sign is equal to the sign of E1.

Example: $E2=5$ and $E3=3$ results in 2, $E2=-5$ and $E3=3$ results in -2

SIGN

When the sign function is used, the sign is returned.

$E1 = \text{sign}(E2)$

Example: $E2=8.9$ results in 1, $E2=0$ results in 0, $E2=-8.9$ results in -1

Note:

1. The E-parameter is stored with the highest accuracy, but the user should be aware that different digits could be stored.

E.g. $E1=99.9999999$ $E3=100.0000001$

$E2=\text{int}(E1)$ gives $E2=99$ $E2=\text{int}(E3)$ gives $E2=100$

Both parameters $E1$ and $E3$ are stored with the same accuracy, the display shows in both cases the value 100, but the result of the function "int" is different.

2. It is advised to add a small value, e.g. the required accuracy of the calculations, to the parameter of which the integer value is to be taken.

E.g. So if $E1=99.9999999$ or $E1=100.0000001$, the expression $E2=\text{int}(E1+.0000001)$ gives $E2=100$ independent of the value from $E1$.

MAXIMUM

The function $\text{max}()$ returns the maximum value of the two arguments.

$E1=\text{max}(E2,E3)$

Example: $E1=\text{max}(16,-10)$ gives $E1=16$

MINIMUM

The function $\text{min}()$ returns the minimum value of the two arguments.

$E1=\text{min}(E2,E3)$

Example: $E1=\text{min}(16,-10)$ gives $E1=-10$

THE CONSTANT PI

The value of pi is stored in the control with an accuracy of 15 digits. At each place, where a value or E-parameter is allowed, the word pi can be used. This constant can be used e.g. with the conversion of angles from radians to decimal degrees or visa versa.

ANGLE IN DECIMAL DEGREES

The default programming mode for an angle is in degrees and decimal parts of a degree. This value can be entered directly in the trigonometric functions, arithmetical or relational expressions.

E.g. $E1 = \sin(44.209303)$

ANGLE IN RADIANS

Sometimes with calculations in which angles are involved, it is useful to express the angle in radians.

360° equal 2π radians.

Therefore an angle of 44.209303° is equal to 0.7715979 radians.

If with the trigonometric functions the angle is in radians, the word rad has to be added to the value, thus:

E.g. $E1=\sin(.7715979 \text{ rad})$

ANGLE CONVERSIONS

Degrees, minutes and seconds to decimal degrees:

An angle of $44^\circ 12' 33.5''$ is converted into a decimal equivalent as follows:

N... $E1=44+12:60+33.5:3600$.

This conversion produces a decimal degree value of $E1=44.209303$.
 Decimal degrees to radians

An angle of $44.209303E$ is converted into radians as follows:

$$N... E1=((44.209303:360)*2*pi)$$

This conversion produces an angle in radians of $E1=.7715979$

Radians to decimal degrees

An angle of 0.771579 radians is converted into decimal degrees as follows:

$$N... E1=(0.771579*360):(2*pi).$$

This conversion produces a decimal degree value of 44.209303 .

TRIGONOMETRICAL FUNCTIONS

The following trigonometric functions are available: sinus (sin), cosine (cos), tangent (tan),
 These are written as:

$$E1=\sin(E2) \quad E1=\cos(E2) \quad E1=\tan(E2)$$

For example, the sine of the angle $44.209303E$ can be programmed in any of the following ways:

$$E1=\sin(44.209303)$$

or

$$E1=\sin(0.7715979 \text{ rad})$$

Notes:

1. Parameter $E2$ represents any arithmetical expression.
2. Odd multiples of $90E$ cannot be used with the tan function; if this occurs an error message is generated.
3. If the angle is in radians, the word rad has to be programmed with the trigonometric functions.

INVERSE TRIGONOMETRICAL FUNCTIONS

The following inverse functions of the trigonometric functions are available:
 arcsin (asin), arccos (acos), arctan (atan).

These are written as:

$$E1=\sin(E2) \quad E1=\cos(E2) \quad E1=\tan(E2)$$

Notes:

1. Parameter $E2$ represents any arithmetical expression.
2. The values of the inverse functions asin and acos should be between -1 and $+1$; atan can have any numerical value.
3. The angle produced by these functions, is in decimal degrees.
4. The angle produced by asin and atan will be between $-90E$ and $+90E$.
5. The angle produced by acos will be between $0E$ and $180E$.

Also possible (V429 and higher):

$$E1=\sin(E3,E4) \quad E1=\cos(E3,E4) \quad E1=\tan(E3,E4) \text{ where } E2=E3:E4$$

Remark: - $\text{abs}(E2)$ must be less than or equal to 1 for acos and asin.
 - the angle created lies between 0° and $+360^\circ$

RELATIONAL EXPRESSIONS

The purpose of a relational expression is to set an E-parameter value to 1 when some conditions are met. If these conditions are not met the value of the parameter is set to 0.

This parameter can be used to perform jumps in the program by means of the G29 function.

The following relations can be used:

equal	=	E1=E2=E3
not equal	<>	E1=E2<>E3
greater	>	E1=E2>E3
greater or equal	>=	E1=E2>=E3
less	<	E1=E2<E3
less or equal	<=	E1=E2<=E3

Example

N.. G29 E1=E2>E3 E1 N=400

This block means:

If parameter E2 is greater than E3, the relation is true and thus parameter E1 set equal 1. Parameter E1 is used in the G29-block as the jump condition. So if E2>E3 a jump to N400 is executed.

Note:

1. Parameters E2 and E3 represent any arithmetical expression.
2. To satisfy a relational expression all digits are compared and have to be the same. When parameter values are produced by calculations, this may cause difficulties. In this case limits have to be set and checks performed to ensure that the value is within these limits.

PRIORITIES OF EVALUATING ARITHMETICAL AND RELATIONAL EXPRESSIONS

Arithmetical and relational operations are performed by the CNC in following order:

1. Evaluate functions: sin, cos, tan, asin, acos, atan, sqrt, abs, int.
2. Calculate reciprocals ($^{-1}$) or perform exponentiation ($^{\wedge}$).
3. Multiply (*) or divide (:).
4. Add (+) or subtract (-).
5. Evaluate relational expressions (=, <>, >, >=, <, <=).

When a block contains operations, which have the same priority, the operations are evaluated from the beginning of the block to the end.

The block E1=3+7:2-4²+5*6 is evaluated in the following order:

1. 4² = 16
2. 7:2 = 3.5
3. 5*6 = 30
4. 3+3.5 = 6.5
5. 6.5 - 16 = -9.5
6. -9.5 + 30 = 20.5

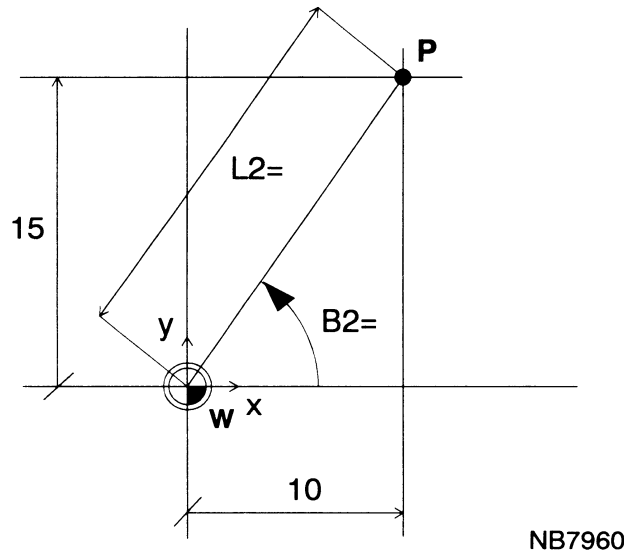
USE OF PARENTHESES ()

Parentheses () can be used to group operations and thus impose a different order of evaluating an expression. The expression between parentheses is evaluated in the normal sequence. Refer to PRIORITIES OF EVALUATING ARITHMETICAL AND RELATIONAL EXPRESSIONS for this sequence. After evaluating the expression the result is used.

A pair of parentheses can be placed within another pair; this is known as 'nesting'. The expression between each pair of parentheses is evaluated starting from the innermost 'nested' pair to the outermost pair. A maximum of four pairs of parentheses can be used in one expression.

Examples

EXAMPLE 1. Calculation of polar coordinates.



NB7960

If the polar coordinates of point P related to the program datum W has to be calculated, the programming could be:

N100 B2=atan(15:10) L2=sqrt(10^2+15^2)

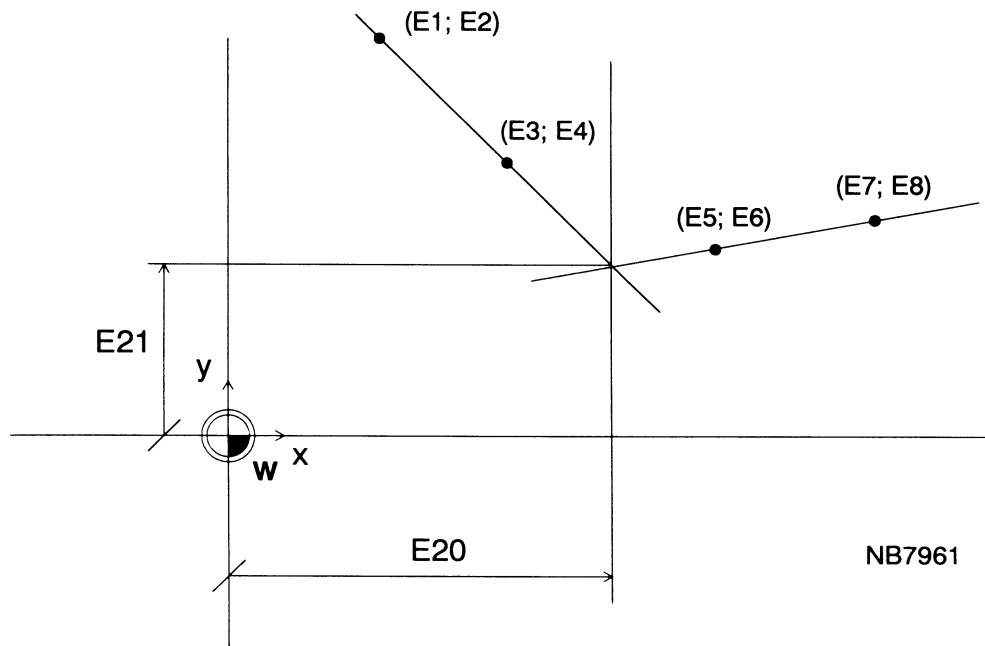
For B2= the calculation is performed in the sequence:

- Calculate 15:10
- Determine angle in decimal degrees

For L2= the calculation is performed in the sequence:

- Calculate 10^2
- Calculate 15^2
- Add 10^2 and 15^2
- Extract the square root.

EXAMPLE 2. Calculation of the intersection point of two lines.



Input parameters

- E1 : first coordinate of first point on first line.
- E2 : second coordinate of first point on first line.

- E3 : first coordinate of second point on first line.
- E4 : second coordinate of second point on first line.

- E5 : first coordinate of first point of second line.
- E6 : second coordinate of first point on second line.

- E7 : first coordinate of second point on second line.
- E8 : second coordinate of second point on second line.

Output parameters

- E20 : first coordinate of intersection point.
- E21 : second coordinate of intersection point.

- E79 =1: an error detected in the macro.
=0: no error.

The macro

N99401 (CALCULATE INTERSECTION POINT TWO LINES)

```

N1  E11=E3-E1 E12=E4-E2 E79=0
N2  E13=sqrt(E11^2+E12^2)
N3  E11=E11:E13 E12=E12:E13
N4  E13=E7-E5 E14=E8-E6
N5  E15=sqrt(E13^2+E14^2)
N6  E13=E13:E15 E14=E14:E15
N7  E16=E11*E13+E12*E14
N8  G29 E15=abs(E16)<.99995 N=12 E15
N9  E79=1
N10 M0 (LINES ARE PARALLEL)
N11 G29 E79 N=17
N12 E15=E1-E7 E16=E2-E8
N13 E17=(E15*E12-E16*E11)
N14 E17=E17:(E13*E12-E11*E14)
N15 E20=E7+E17*E13
N16 E21 =E8+E17*E14
N17

```

Explanation:

N1-N3: the unit vector of the first line is calculated.
 N4-N6: the unit vector of the second line.
 N7-N8: check to see if the unit vectors are not parallel.
 N9-N11: if the lines are parallel, parameter E79 is set an error displayed with a program stop.
 After the start the calculations are not performed.
 N12-N14: compute the factor of the vector.
 N15-N16: calculate the coordinates of the intersection point.

Remark:

Parameter E79 can be used to handle the error in the activating program or macro.

Example of how the macro is used

First line through the points (30,50) and (60,30).

Second line through the points (100,50) and (50,10).

The calculation of the intersection point could be programmed as:

```

N100 E1=30 E2=50 E3=60 E4=30
N101 E5=100 E6=50 E7=50 E8=10
N102 G22 N =99401
N103 G29 E79 K0 N=...
N104 G0 X=E20 Y=E21

```

Explanation:

N100: the points on the first line.
 N101: the points of the second line.
 N102: calculation of the intersection point.
 N103: if an error is detected, transfer control to block, which contains M30.
 N104: move with rapid traverse to the intersection point.

12. Turning mode

12.1 Introduction

The turning mode has been developed for machines with a C axis that can turn continuously. In this way, turning operations can be carried out on a milling machine.

The C axis can be switched to turning mode. The C axis is then programmed as a turning spindle via S1= and M1. The turning tools are mounted in the milling spindle and clamped at the desired orientation.

In special cases, the milling spindle can be programmed parallel to the turning spindle via S and M. A second milling spindle is not possible on machines with turning mode.

Notes and application

AVAILABILITY

Machine and CNC must be prepared for turning mode by the machine builder. If your machine is not equipped with all the G functions described here, please refer to your machine manual.

GRAPHICS

The graphic is not displayed symmetrically to the rotation

DISPLAY

If G36 is active, the display of the C axis position changes to display S1=. The machining status is expanded with G36/G37.

The spindle power display shows the power of the second spindle even if both spindles are active.

REFERENCE POINT

When the controller runs up, it is always in milling mode G37. The C axis can only be switched to turning mode after the reference points have been approached.

ZERO POINT

In turning mode, the workpiece zero point in X should lie in the centre of rotation of the S1 axis. It is recommended that the workpiece zero point in Y should also lie in the centre of rotation of the S1 axis.

SPINDLE OVERRIDE

Spindle override is effective for both spindles in turning mode (G36).

12.2 Machine constants

New machine constants

MC 314 Turning mode (0=off, 1=on)

Activated:

- G functions G36 and G37
- Turning cycles
- Machine constants MC2600 - MC27xx, MC45xx

MC 450 Balancing: measurement axis (1=X, 2=Y, 3=Z)

This MC determines the axis on which the rotary table is installed. Unbalance is easiest to measure in this axis. Normally, 2 = Y axis.

The MC is used in the 'unbalance calibration' (installation), G691 'unbalance detection' and G692 'unbalance checking' cycles.

MC 451 Balancing: maximum amplitude [μm].

This MC specifies the permissible residual amplitude in the measuring axis. The measurement is cancelled if the measured amplitude is greater than MC451 at a particular speed. Normally 5 μm .

The MC is used in the 'unbalance calibration' (installation), G691 'unbalance detection' and G692 'unbalance checking' cycles. The C1 parameter can be superimposed on this in the G691 and G692 cycles.

MC 452 Balancing: initial radial position [μm].

This MC specifies the radial position (distance from centre point) of the rotary table (S1 axis) at which a balancing mass is normally mounted to compensate for unbalance.

The MC is used in the G691 'unbalance detection' cycle.

MC 453 Balancing: rotary table displacement [mGrad].

This MC specifies the 0 position of the rotary table and the position (door) where the operator fits the mass to compensate (and calibrate) the unbalance.

The MC is used in the 'unbalance calibration' (installation) and G691 'unbalance detection' cycles.

Existing machine constants.

MC 268 second spindle (0=no, 1=yes).

MC2600 - MC2799, MC4500 - MC4599 second spindle.

Remark: The inverter output of the 2nd spindle (MC2600) should be equal to the inverter output of the C axis (MC3400).

12.3 G36/G37 Activate/deactivate turning mode

Purpose

- | | |
|-----|---|
| G36 | Switches the machine from milling mode on the C axis to turning mode with turning spindle S1. |
| G37 | Terminates turning mode. Switches the machine back to milling |

Format

N... G36 or N... G36

Parameter

none

Type of function

modal

Notes and application

G36

The CNC switches the C axis to turning mode.

In turning mode, the circular axis is programmed as a second spindle using S1= and M1. C parameters can no longer be programmed.

The display of C (setpoint and actual value) on the screen is switched to S1. If the turning spindle is stationary, the position (0-359.999 degrees) is displayed.

G95 is active, assigned to the second spindle.

All G functions can be programmed, but not all the G functions are meaningful. For instance, a pocket has no meaning in turning mode. The C parameters and certain other parameters can no longer be programmed in certain G functions.

The effect of G36 remains active until it is cancelled by G37, runup or <CNC reset>. G36 is not cancelled by M30 or <Cancel program>.

G37

The CNC switches the C axis on again.

If the rotary spindle is still turning at the start of G37, it is first stopped.

The position of the circular axis is displayed on the screen with a value between 0 and 359.999 degrees.

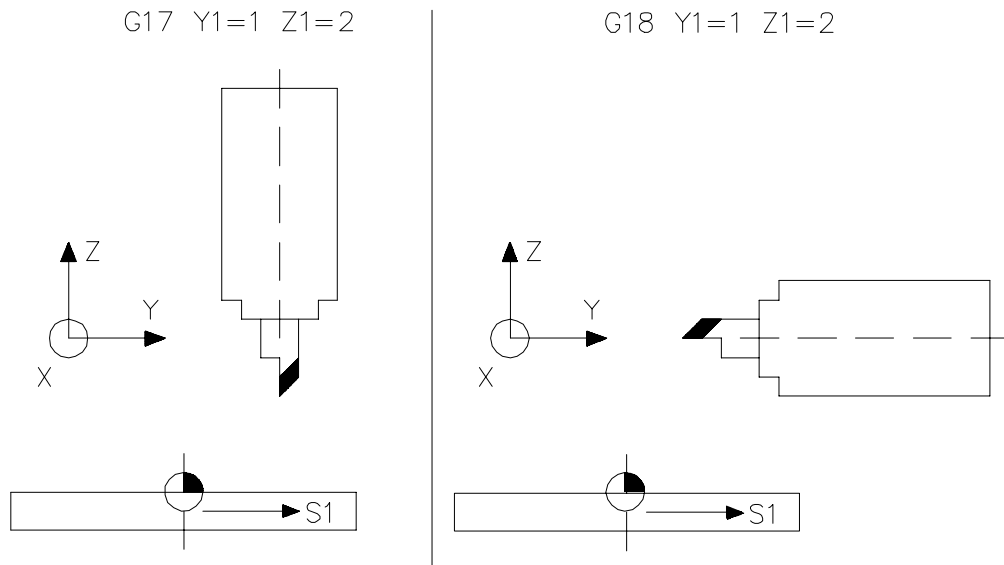
G94 becomes active.

The effect of G37 remains active until it is cancelled by G36. G37 is not cancelled by M30 or <Cancel program>. G27 is always active following runup or <CNC reset>.

12.4 G17 Mainplane for turning mode (G17 Y1=1 Z1=2)

In turning mode, the machine is in G17 or G18. This determines the direction of tool length correction (Z direction in G17, Y direction in G18).

In turning mode, machining takes place in the YZ plane. These planes are automatically taken into account in the turning cycles. The special machining plane is indicated in G17 by Y1=1 (first main axis) and Z1=2 (second main axis).



Angles (positive) and circular directions (CW) are defined from the Y axis to the Z axis.

Remark: The special plane must be reset at the end of turning mode by programming a 'normal' G17 or G18 without addresses.

12.5 G94/G95 Expanded choice of feed unit

Purpose

Informs the CNC how to evaluate the programmed speed (S).

This function is expanded for turning mode.

The spindle and the circular table must be programmed for turning.

Notes and application

In addition, the rotary table (second spindle) must be programmed with S1= and M1= for turning.

In milling mode (G37): N... G95 F.. {S..} {M..}

In turning mode (G36): N... G95 F.. {S1=..} {M1=..}

S and M refer to the spindle

S1= and M1= refer to the second spindle

PRIORITY

The active spindle speed is either S or S1=. If S and S1= are both programmed, S1 is used.

MAXIMUM SPEED

The value of the second spindle speed (S1=) lies between 0 and 'Max. output voltage speed' (MC2691).

MACHINE FUNCTION

Second spindle machine functions:

M1=3 second spindle clockwise

M1=4 second spindle anticlockwise

M1=5 second spindle stop

Positioning of the second spindle (M1=19) is not possible. Positioning takes place in milling mode.

The S1= and M1= addresses can also be programmed in the following G functions: G0, G1, G2, G3, G94.

The G95 function calculates the feed in [mm/min (inches/min)] based on the programmed feed in [mm/rev], [inches/rev] and the active spindle speed.

12.6 G96/G97 Constant cutting speed

Purpose

G96 Programming constant cutting speed.
G97 Switching off constant cutting speed.

Format

N... G96 F.. D.. {S..} {M..} {S1=..} {M1=..}
N... G97 F.. {S..} {M..} {S1=..} {M1=..}

Parameter

G Constant cutting speed
D Upper speed limit (rev/min)
F Feed
S Cutting speed (m(feet)/min)
M Machine function
S1= Cutting speed (m(feet)/min)
M1= Machine function

G96

G Spindle speed
S Speed (rev/min)
M Machine function
S1= Speed (rev/min)
M1= Machine function

G97

S and M refer to the spindle
S1= and M1= refer to the second spindle (rotary table)

Type of function

modal

Notes and application

MAXIMUM SPEED (D)

The value of the second spindle speed lies between 0 and 'Max. output voltage speed' (MC2691).

MACHINE FUNCTION

Second spindle machine functions:

M1=3 second spindle clockwise
M1=4 second spindle anticlockwise.
M1=5 second spindle stop

Positioning of the second spindle (M1=19) is not possible. Positioning takes place in milling mode.

The G96 function calculates the feed in [mm/min (inches/min)] based on the programmed feed in [mm/rev], [inches/rev] and the active spindle speed.

The active spindle speed is either S or S1=. If S and S1= are both programmed, S1 is used.

12.7 Defining turning tools in the tool table

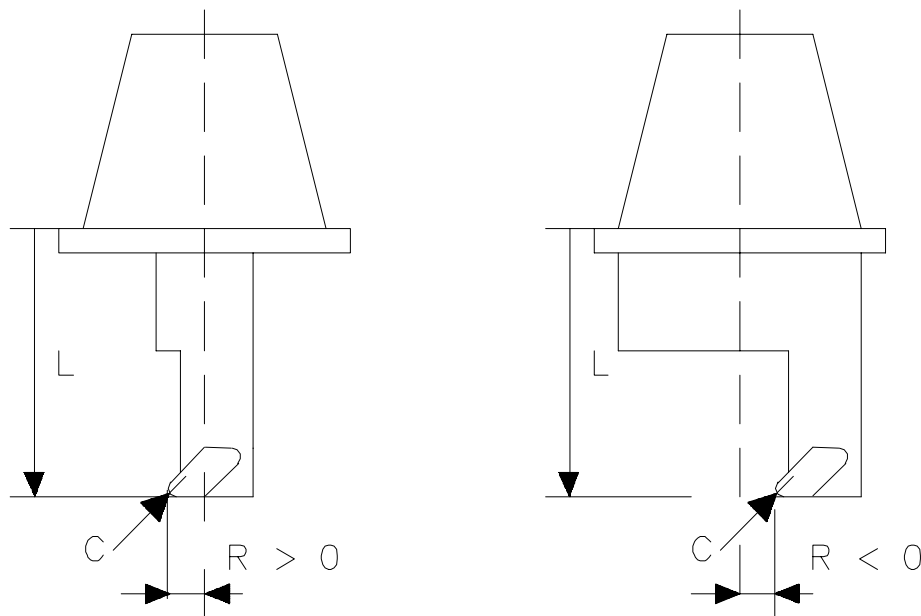
Tool correction and orientation

The tool dimensions are stored as tool length L, tool radius R and tool corner radius C. The tool radius correction refers to the corner radius C and the necessary tool orientation is entered in address O in the tool memory.

Tool memory

The turning tools can be placed in any position in the tool magazine. The tool is marked as a turning tool by parameter Q3= 'tool type'. This also locks the spindle.

Q3= tool type' = 8xx turning tool.



Tool dimensions for turning tools.

If turning mode is active (or in general, when the main plane is parallel to the tool axis), the radius R is regarded as a displacement. The radius correction is also calculated in this case using the tool corner radius C and the orientation O. If the orientation is in the negative direction, the tool radius is also calculated as a negative displacement.

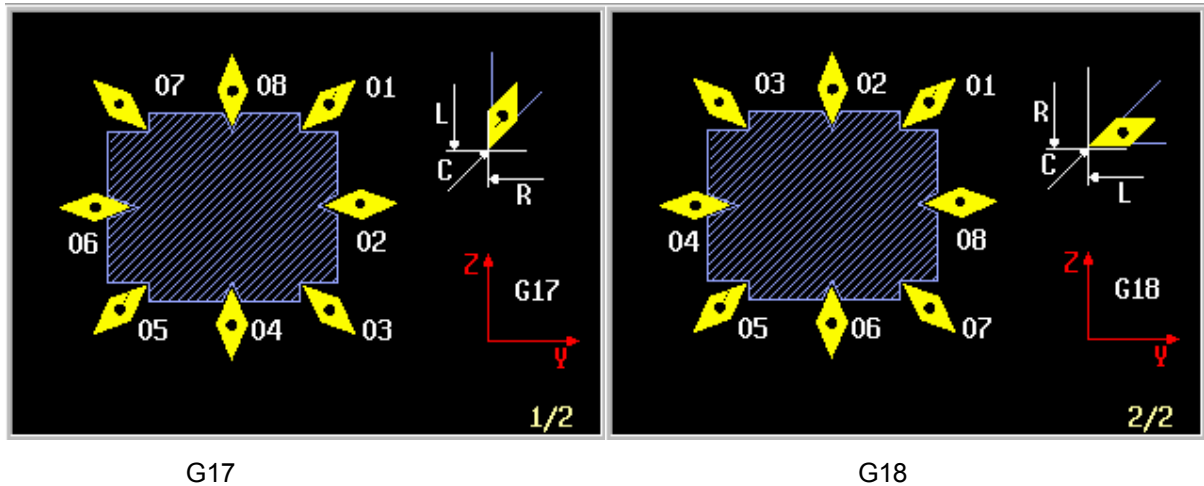
Plane	Orientation	Radius correction	Radius as displacement
G17	not active	R	not active
G17 Y1=1 Z1=2	1, 2, 3, 4, 8	C and O	R in negative Y- direction
G17 Y1=1 Z1=2	5, 6, 7	C and O	R in positive Y- direction
G18	not active	R	not active
G18 Y1=1 Z1=2	1, 2, 6, 7, 8	C and O	R in negative Z- direction
G18 Y1=1 Z1=2	3, 4, 5	C and O	R in positive Z- direction

The orientation must be defined in the tool memory for G17 Y1=1 Z1=2 (vertical operation). The controller adapts the active orientation when G18 Y1=1 Z1=2 (horizontal operation) is activated.

12.8 G302 Overwriting radius compensation parameters

Purpose

The G302 function determines the tool orientation during execution. The tool parameters in the tool memory are not changed.



Format

N.. G302 O

Parameter

G	Override radius comp. parameters
0	Tool Orientation

- O Defines the tool orientation used during execution. The value lies between 0 and 8.

Type of function

non modal

Notes and application**Remarks:**

If the active tool orientation is overwritten, the direction of the R displacement may also change.

In G18, the active tool orientation is already changed by the CNC. See chapter 'Tool correction'.

USE

The G302 function should be used if, for example, the main spindle has been turned through 180 degrees with M19 D90. In this case, the orientation is mirrored compared with the status with M19 D90. The orientation should also be mirrored when turning takes place 'across the centre'.

Remark: In these cases, the direction of rotation of the 2nd spindle should also be reversed.

DELETING

G302 is switched off again with G302 without parameter, set plane (G17, G18, G19), tool change, M30 and <Cancel program>.

12.9 Unbalance cycles

12.9.1 General information

To machine workpieces to be turned on an FP machine, both the machine (rotary table) and workpiece must be balanced, otherwise the life of the machine, the quality of the workpiece or even the safety of the operator cannot be guaranteed.

First, the unbalance properties of the rotary table must be determined. Usually, this unbalance calibration takes place when the machine is handed over or during servicing.

To determine the unbalance of the clamped workpiece, a new cycle has been introduced: **G691 unbalance detection**.

This cycle can be called up directly in manual mode under the FST menu.

The result is a suggestion for compensating for the measured unbalance: what mass should be attached at what radial position from the turning centre. The rotary table is automatically turned to the position where the mass should be attached.

The radial position for an available compensating mass can be calculated in the dialog window. The relationship between mass and position are shown graphically.

To ensure that no turning operations take place in automatic mode with too great an unbalance, a new G function can be called in the program: **G692 unbalance check**.

This G function checks the unbalance present against the permissible unbalance. If this is exceeded, an error message is issued, following which the operator can cancel the automatic mode and carry out a new unbalance detection with correction in manual mode.

12.9.2 Description of unbalance

When working in turning mode, centrifugal forces occur if the clamped part (e.g. a pump housing) has an unbalance. This influences concentric accuracy because the second spindle (= circular axis C) is configured on the Y axis.

Unbalance $U = m \cdot R$

where:

m	= mass	[g]
R	= distance from centre of mass to centre of table	[mm]

The unbalance is given in [gmm] (grammes*mm). This means that 500 [grammes] at 300 [mm] (=150000 [gmm]) has the same effect as 1000 [grammes] at 150 [mm].

The centrifugal force is proportional to the unbalance and rises quadratically with rising speed.

Centrifugal force $F_c = m \cdot R : 1000000 \cdot (S : 2 \cdot \pi : 60)^2$

where:

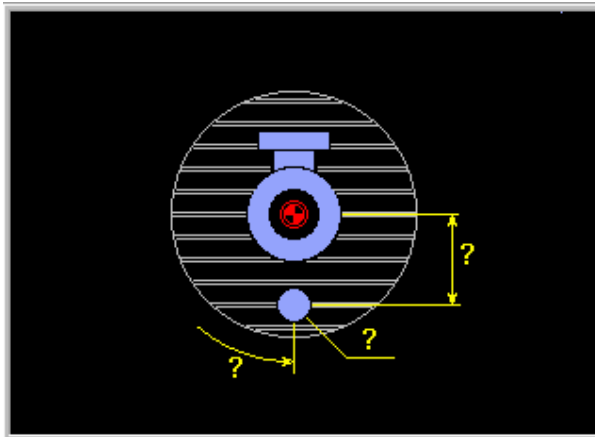
F _c	= centrifugal force	[N]
m	= mass	[g]
R	= distance from centre of mass to centre of table	[mm]
S	= speed	[rpm]

The unbalance must be compensated by a balance weight. For this purpose, the available measuring systems of the circular axis C and the linear axis Y are used to detect the unbalance that exists.

12.9.3 G691 Unbalance measurement

Purpose

This cycle calculates the instantaneous unbalance. It gives the operator a suggestion how to compensate for the unbalance. This cycle should be called after each clamping operation and after milling mode.



Format

N.. G691 {D..}

Parameter

G	Unbalance measurement	
D	Speed limitation	[rev/min]

- D** Maximum speed for terminating the measurement
 Default setting MC2691 'maximum speed'
 Minimum value 50 [rpm]
 The speed limit should be at least as high as the programmed speed for turning machining.

Type of function

Non modal

Associated functions

G692

Notes and application

When detecting unbalance, the position error of the linear axis is measured with rising speed. The speed is increased in steps of 25 rpm. When the position error has reached the maximum value (MC451) or the maximum speed has been reached, the measurement is terminated. The unbalance is calculated from the measured error and the stored calibration data.

The unbalance (gmm) and compensation position (degrees) are displayed. This position is approached at the end of the cycle.

Example: Balancing a workpiece

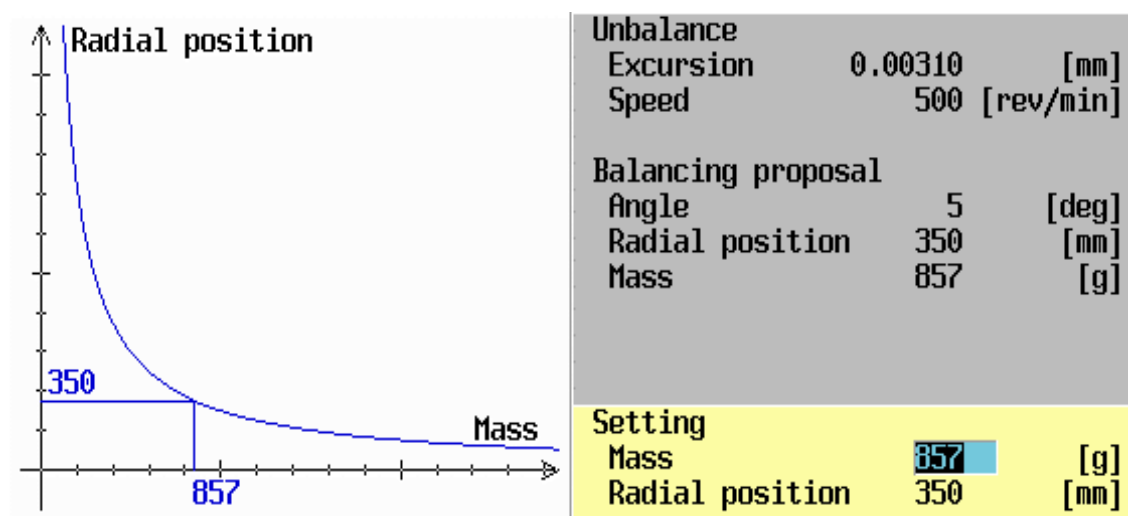
G691 D500

Explanation:

- 1 Start balancing cycle with maximum speed of 500 rpm.
- 2 Unbalance is measured. Calculated mass and radial position (distance and angle) are shown in the window. The balance position is automatically positioned.
- 3 Enter the weight of an available mass in the window.
- 4 The CNC displays the new radial distance for the available mass.
- 5 Fit the mass at the radial position (distance and angle). Terminate with start.
- 6 Check the balance quality by repeating the balancing cycle G691. The unbalance mass must be very small. If necessary, balance again with the displayed mass.

Representation of measurement results

Once the unbalance detection measurement is terminated, the measurement results are displayed instead of the input and support fields. This image is created by G350.



Left:

The relationship between mass and position are shown graphically.

Top right:

The measured unbalance causes a deflection at the speed displayed. This unbalance can be compensated in accordance with the balancing suggestion.

Bottom right:

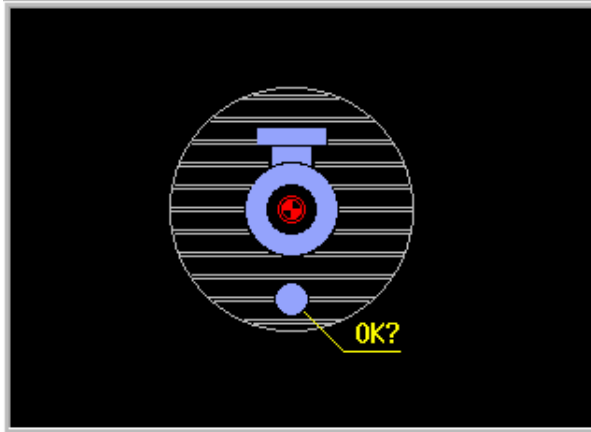
The radial position for a selected mass is calculated in the dialog window. The calculation takes place after pressing the <ENTER> key. The START key terminates the cycle and closes this window.

In **automatic mode**, the left graphical window is not shown so that the program pointer remains visible.

12.9.4 G692 Unbalance check

Purpose

This cycle checks that the unbalance does not exceed a particular value. It should be called at the start of every turning operation to ensure that the concentric error does not exceed the desired tolerance or the specified limit.



Format

N... G692 {C1=..} {D..}

Parameter

G	Unbalance check	
C1=	Allowed excursion	[mm inch]
D	Check speed	[rev/min]

- C1= Maximum unbalance for message
Default setting MC451 "maximum deflection"
- D Programmed speed for checking
Default setting MC2691 "maximum speed"

Type of function

modal

Associated functions

G691

Notes and application

When checking unbalance, the deflection of the linear axis is measured at a specified speed. If the deflection reaches the value C1=, an error message is issued.

Example: Checking unbalance.

G692 C1=0.003 D500

Explanation:

The CNC detects whether the deflection of the table is within the limit of 0.003 mm at a speed of 500 rpm. If the deflection is greater than the value entered (C1=), the program is stopped.

12.9.5 Unbalance example

N1 G691 D500

N2 G691 D500

N... milling

N30 G37

N31 G692 D500

N... turning

Explanation:

N1 G691 D500

1 Start balancing cycle with maximum speed of 500 rpm.

2 Unbalance is measured. Calculated mass and radial position (distance and angle) are shown in the window. The balance position is automatically located.

3 Enter the weight of an available mass in the window.

4 The CNC displays the new radial distance for the available mass.

5 Fit the mass at the radial position (distance and angle). Terminate with start.

N2 G691 D500

Check the balance quality by repeating the balancing cycle G691. The unbalance mass must be very small. If necessary, balance again with the displayed mass.

N...milling

Unbalance may change due to milling processes or changes in the clamping.

N30 G37

Start turning mode

N31 G692 D500

Check whether unbalance is still correct.

12.10 Turning cycles

AVAILABILITY

Machine and CNC must be prepared for turning operations by the machine builder. If your machine is not equipped with all the G functions described here, please refer to your machine manual.

The turning cycles are executed as macros, every block can be seen in the display and each block is processed as a single block.

General Notes and application

STARTING POINT

The starting point determines the place where the tool starts machining. The cutting steps start from this position. If the tool is a long distance away, several cutting steps take place. If the tool is between Y1= and Y2=, cutting will start there and the cutting may not all be carried out.

If the co-ordinate of the starting point Y is smaller than the co-ordinate of the machining starting point Y1, the machine first travels to co-ordinate Z1.

TOOL MEMORY ADDRESSES

The following addresses are used in the tool memory:

C	Tool tip radius
O	Tool orientation

If no O is entered in the tool memory, a standard orientation is assumed depending on the machining.

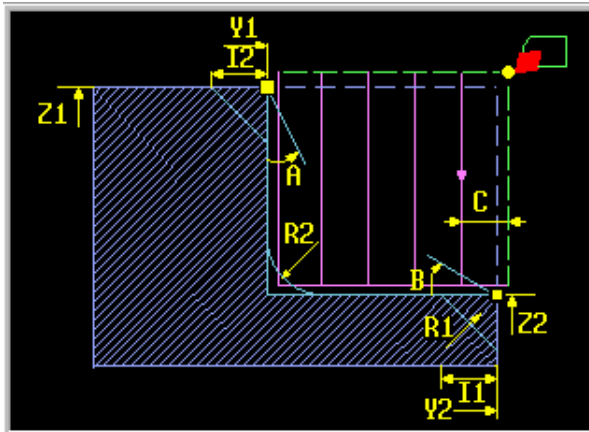
RADIUS COMPENSATION

Tool tip radius compensation is carried out automatically in this G function.

12.10.1 G822 Clearance axial

Purpose

Clearance axial



Format

N... G822 Y.. Z.. Y1=.. Z1=.. Y2=.. Z2=.. C.. {A..} {B..} [{I1=..}{R1=..}] [{I2=..}{R2=..}] {S1=..} {F}

Parameter

G Clearance axial (E)
 Y Startpoint
 Z Startpoint
 Y1= Beginpoint contour
 Z1= Beginpoint contour
 Y2= Endpoint contour
 Z2= Endpoint contour
 C Cutting depth
 A (Angle 1)
 B (Angle 2)
 I1= (Chamfer length 1)
 R1= (Radius 1)
 I2= (Chamfer length 2)
 R2= (Radius 2)

S1= ((Cutting) Speed)
 F (Feed)

- Y Starting point. (Recommended value: safety distance up to Y2=)
 Position of tool in radial direction. This position is the starting point for machining. Y is reduced with C until Y1= is reached.
- Z Starting point. (Recommended value: safety distance up to Z1=)
 Position of tool in axial direction. This position is the starting point for machining. Machining starts at Z until Z2 is reached.
- Y1= Contour starting point
 Starting point of the contour to be machined.
- Z1= Contour starting point
 Starting point of the contour to be machined.
- Y2= Contour end point
 End point of the contour to be machined.

- Z2= Contour end point
End point of the contour to be machined.
- C Radial feed depth (incremental):
Dimension by which the tool is fed in the radial direction in each case. The depth does not have to be a multiple of the feed depth.
- A Angle: Basic setting A=0.
Angle (>0) at contour starting point. Angle A or B must be chosen so that the tool does not undercut.
- B Angle: Basic setting B=0.
Angle (>0) at contour end point.
- I1= Chamfer length: Basic setting I1=0.
Chamfer length at contour end point.
Only I1= or R1= may be programmed.
- R1= Rounding: Basic setting R1=0.
Rounding at contour end point.
- I2= Chamfer length: Basic setting I2=0.
Chamfer length at contour starting point.
- R2= Rounding: Basic setting R2= tool tip radius.
Rounding between angles A and B.
- F Feed rate: Traverse speed of tool in mm/min
- S1= Spindle speed

Type of function

non modal

Associated functions

G827 for finishing

Notes and application

Cutting takes place first, then finishing.

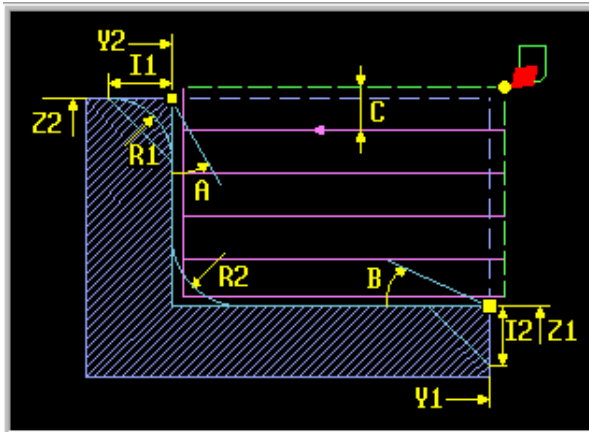
Tool orientation may only be 4, 5 or 6.

The tool path is corrected for the tip radius.

12.10.2 G823 Clearance radial

Purpose

Clearance radial



Format

N... G823 Y.. Z.. Y1=.. Z1=.. Y2=.. Z2=.. C.. {A..} {B..} [{I1=..}{R1=..}] [{I2=..}{R2=..}] {S1=..} {F}

Parameter

G Clearance radial (E)
 Y Startpoint
 Z Startpoint
 Y1= Beginpoint contour
 Z1= Beginpoint contour
 Y2= Endpoint contour
 Z2= Endpoint contour
 C Cutting depth
 A (Angle 1)
 B (Angle 2)
 I1= (Chamfer length 1)
 R1= (Radius 1)
 I2= (Chamfer length 2)
 R2= (Radius 2)

S1= ((Cutting) Speed)
 F (Feed)

- Y Starting point. (Recommended value: safety distance up to Y1=)
 Position of tool in radial direction. This position is the starting point for machining. Machining starts at Y until Y2 is reached.
- Z Starting point. (Recommended value:safety distance up to Z2=)
 Position of tool in axial direction. This position is the starting point for machining. Z is reduced with C until Z1= is reached.
- Y1= Contour starting point
 Starting point of the contour to be machined.
- Z1= Contour starting point
 Starting point of the contour to be machined.
- Y2= Contour end point
 End point of the contour to be machined.

TURNING MODE

- Z2= Contour end point
End point of the contour to be machined.
- C Radial feed depth (incremental):
Dimension by which the tool is fed in the axial direction in each case. The depth does not have to be a multiple of the feed depth.
- A Angle: Basic setting A=0.
Angle (>0) at contour starting point.
Angle A or B must be chosen so that the tool does not undercut.
- B Angle: Basic setting B=0.
Angle (>0) at contour end point.
- I1= Chamfer length: Basic setting I1=0.
Chamfer length at contour end point.
Only I1= or R1= may be programmed.
- R1= Rounding: Basic setting R1=0.
Rounding at contour end point.
- I2= Chamfer length: Basic setting I2=0.
Chamfer length at contour starting point.
- R2= Rounding: Basic setting R2= tool tip radius.
Rounding between angles A and B.
- F Feed rate: Traverse speed of tool in mm/min.
- S1 Spindle speed

Type of function

non modal

Associated functions

G827 for finishing

Notes and application

Cutting takes place first, then finishing.

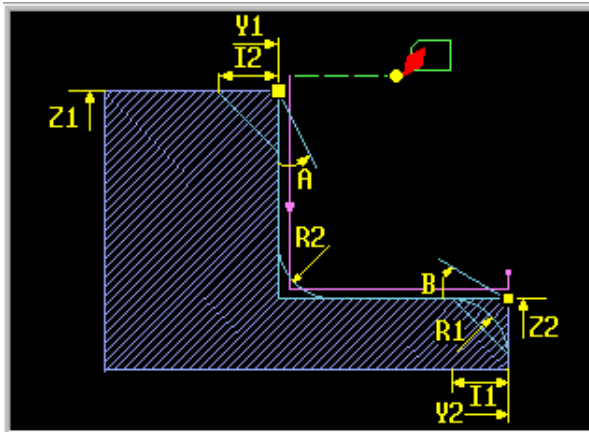
Tool orientation may only be 4, 5 or 6.

The tool path is corrected for the tip radius.

12.10.3 G826 Clearance axial finishing

Purpose

Clearance axial finishing



Format

N... G826 Y.. Z.. Y1=.. Z1=.. Y2=.. Z2=.. {A..} {B..} [{I1=..}]{R1=..} [{I2=..}]{R2=..} {S1=..} {F}

Parameter

G Clearance axial finishing (E)
 Y Startpoint
 Z Startpoint
 Y1= Beginpoint contour
 Z1= Beginpoint contour
 Y2= Endpoint contour
 Z2= Endpoint contour
 A (Angle 1)
 B (Angle 2)
 I1= (Chamfer length 1)
 R1= (Radius 1)
 I2= (Chamfer length 2)
 R2= (Radius 2)
 S1= ((Cutting) Speed)

F (Feed)

- Y Starting point. (Recommended value: safety distance up to Y1=)
 Position of tool in radial direction. This position is the starting point for finishing.
- Z Starting point. (Recommended value: safety distance up to Z1=)
 Position of tool in axial direction. This position is the starting point for finishing.
 Finishing starts at Y.
- Y1= Contour starting point
 Starting point of the contour to be machined.
- Z1= Contour starting point
 Starting point of the contour to be machined.
- Y2= Contour end point
 End point of the contour to be machined.

TURNING MODE

- Z2= Contour end point
End point of the contour to be machined.
- A Angle: Basic setting A=0.
Angle (>0) at contour starting point.
Angle A or B must be chosen so that the tool does not undercut.
- B Angle: Basic setting B=0.
Angle (>0) at contour end point.
- I1= Chamfer length: Basic setting I1=0.
Chamfer length at contour end point.
Only I1= or R1= may be programmed.
- R1= Rounding: Basic setting R1=0.
Rounding at contour end point.
- I2= Chamfer length: Basic setting I2=0.
Chamfer length at contour starting point.
- R2= Rounding: Basic setting R2= tool tip radius.
Rounding between angles A and B.
- F Feed rate: Traverse speed of tool in mm/min
- S1= Spindle speed

Type of function

non modal

Associated functions

G822 for roughing

Notes and application

Finishing goes from Y1/Z1 to Y2/Z2.

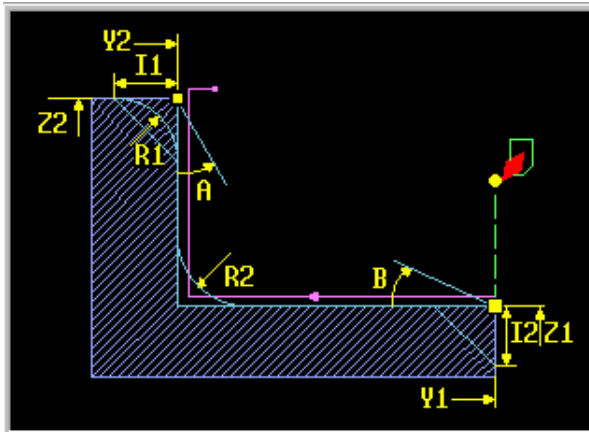
Tool orientation may only be 4, 5 or 6.

The tool path is corrected for the tip radius.

12.10.4 G827 Clearance radial finishing

Purpose

Clearance radial finishing



Format

N... G827 Y.. Z.. Y1=.. Z1=.. Y2=.. Z2=.. {A..} {B..} [{I1=..}]{R1=..} [{I2=..}]{R2=..} {S1=..} {F}

Parameter

G Clearance radial finishing (E)
 Y Startpoint
 Z Startpoint
 Y1= Beginpoint contour
 Z1= Beginpoint contour
 Y2= Endpoint contour
 Z2= Endpoint contour
 A (Angle 1)
 B (Angle 2)
 I1= (Chamfer length 1)
 R1= (Radius 1)
 I2= (Chamfer length 2)
 R2= (Radius 2)
 S1= ((Cutting) Speed)

F (Feed)

- Y Starting point. (Recommended value: safety distance up to Y1=)
 Position of tool in radial direction. This position is the starting point for finishing.
 Finishing starts at Y until Y2 is reached.
- Z Starting point. (Recommended value: safety distance up to Z2=)
 Position of tool in axial direction. This position is the starting point for finishing.
- Y1= Contour starting point
 Starting point of the contour to be machined.
- Z1= Contour starting point
 Starting point of the contour to be machined.
- Y2= Contour end point
 End point of the contour to be machined.
- Z2= Contour end point

End point of the contour to be machined.

- A Angle: Basic setting A=0.
 Angle (>0) at contour starting point.
 Angle A or B must be chosen so that the tool does not undercut.
- B Angle: Basic setting B=0.
 Angle (>0) at contour end point.
- I1= Chamfer length: Basic setting I1=0.
 Chamfer length at contour end point.
 Only I1= or R1= may be programmed.
- R1= Rounding: Basic setting R1=0.
 Rounding at contour end point.
- I2= Chamfer length: Basic setting I2=0.
 Chamfer length at contour starting point.
- R2= Rounding: Basic setting R2= tool tip radius.
 Rounding between angles A and B.
- F Feed rate: Traverse speed of tool in mm/min
- S1 Spindle speed

Type of function

non modal

Associated functions

G823 for roughing

Notes and application

Finishing goes from Y1/Z1 to Y2/Z2.

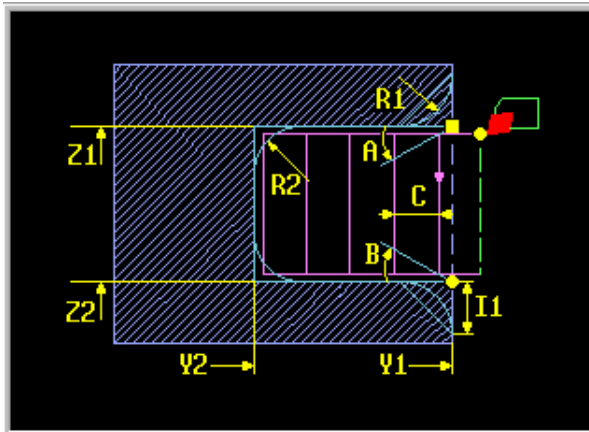
Tool orientation may only be 4, 5 or 6.

The tool path is corrected for the tip radius.

12.10.5 G832 Roughing axial

Purpose

Roughing axial



Format

N... G832 Y.. Z.. Y1=.. Z1=.. Y2=.. Z2=.. C.. {A..} {B..} [{I1=..}]{R1=..} {R2=..} {S1=..} {F}

Parameter

G Roughing axial (E)
 Y Startpoint
 Z Startpoint
 Y1= Beginpoint contour
 Z1= Beginpoint contour
 Y2= Endpoint contour
 Z2= Endpoint contour
 C Cutting depth
 A (Angle 1)
 B (Angle 2)
 I1= (Chamfer length 1)
 R1= (Radius 1)
 R2= (Radius 2)
 S1= ((Cutting) Speed)

F (Feed)

- Y Starting point. (Recommended value: safety distance up to Y1=)
 Position of tool in radial direction. This position is the starting point for machining.
 Machining starts at Y and is reduced with C until Y2= is reached.
- Z Starting point. (Recommended value: safety distance up to Z1=)
 Position of tool in axial direction. This position is the starting point for machining.
 Machining starts at Z1= until Z2= is reached.
- Y1= Contour starting point
 Starting point of the contour to be machined.
- Z1= Contour starting point
 Starting point of the contour to be machined.
- Y2= Contour end point
 End point of the contour to be machined.

TURNING MODE

- Z2= Contour end point
End point of the contour to be machined.
- C Radial feed depth (incremental):
Dimension by which the tool is fed in the radial direction in each case. The depth does not have to be a multiple of the feed depth.
- A Angle: Basic setting A=0.
Angle (>0) at contour starting point. (Z1=)
Angles A and B must be chosen so that the tool does not undercut.
- B Angle: Basic setting B=0.
Angle (>0) at contour end point. (
- I1= Chamfer length: Basic setting I1=0.
Chamfer length at start and end of contour.
Only I1= or R1= may be programmed.
- R1= Rounding: Basic setting R1=0.
Rounding at start and end of contour.
- R2= Rounding: Basic setting R2= tool tip radius.
Rounding at the bottom of the contour.
- F Feed rate: Traverse speed of tool in mm/min
- S1= Spindle speed

Type of function

non modal

Associated functions

G837 for finishing

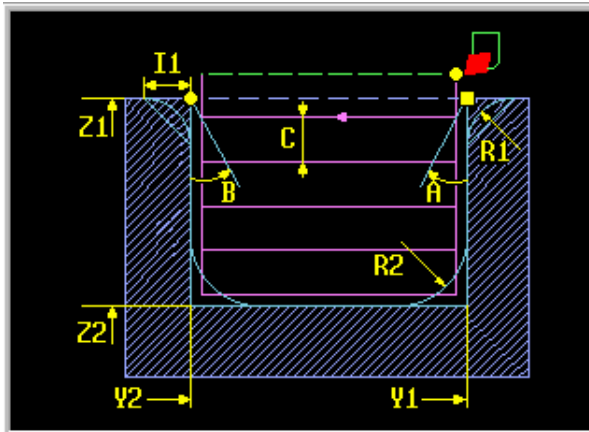
Notes and application

Rough cutting takes place first, then finishing.
Tool orientation may only be 3, 4 or 5.
The tool path is corrected for the tip radius.

12.10.6 G833 Roughing radial

Purpose

Roughing radial



Format

N... G833 Y.. Z.. Y1=.. Z1=.. Y2=.. Z2=.. C.. {A..} {B..} [{I1=..}]{R1=..} {R2=..} {S1=..} {F}

Parameter

G Roughing radial (E)
 Y Startpoint
 Z Startpoint
 Y1= Beginpoint contour
 Z1= Beginpoint contour
 Y2= Endpoint contour
 Z2= Endpoint contour
 C Cutting depth
 A (Angle 1)
 B (Angle 2)
 I1= (Chamfer length 1)
 R1= (Radius 1)
 R2= (Radius 2)
 S1= ((Cutting) Speed)

F (Feed)

- Y Starting point. (Recommended value: safety distance up to Y1=)
 Position of tool in radial direction. This position is the starting point for machining.
 Machining starts at Y1= until Y2= is reached.
- Z Starting point. (Recommended value: safety distance up to Z1=)
 Position of tool in radial direction. This position is the starting point for machining.
 Machining starts at Z and is reduced with C until Z2= is reached.
- Y1= Contour starting point
 Starting point of the contour to be machined.
- Z1= Contour starting point
 Starting point of the contour to be machined.
- Y2= Contour end point

End point of the contour to be machined.

- Z2= Contour end point
End point of the contour to be machined.
- C Radial feed depth (incremental):
Dimension by which the tool is fed in the axial direction in each case. The depth does not have to be a multiple of the feed depth.
- A Angle: Basic setting A=0.
Angle (>0) at contour starting point. (
Angles A and B must be chosen so that the tool does not undercut.
- B Angle: Basic setting B=0.
Angle (>0) at contour end point. (Y2=)
- I1= Chamfer length: Basic setting I1=0.
Chamfer length at start and end of contour.
Only I1= or R1= may be programmed.
- R1= Rounding: Basic setting R1=0.
Rounding at start and end of contour.
- R2= Rounding: Basic setting R2= tool tip radius.
Rounding at the bottom of the contour.
- F Feed rate: Traverse speed of tool in mm/min
- S1 Spindle speed

Type of function

non modal

Associated functions

G837 for finishing

Notes and application

Rough cutting takes place first, then finishing.

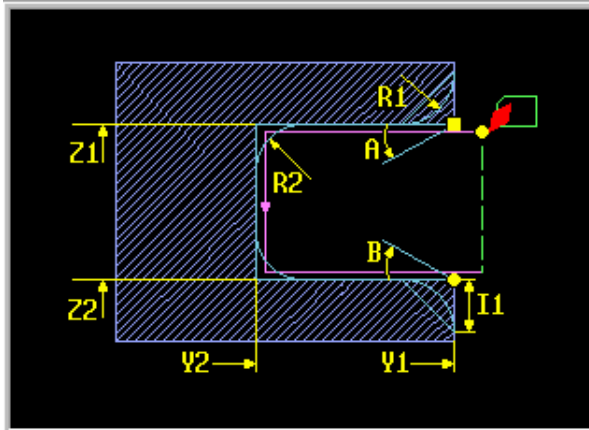
Tool orientation may only be 5, 6 or 7.

The tool path is corrected for the tip radius.

12.10.7 G836 Roughing axial finishing

Purpose

Roughing axial finishing



Format

N... G836 Y.. Z.. Y1=.. Z1=.. Y2=.. Z2=.. {A..} {B..} [{I1=..}]{R1=..} {R2=..} {S1=..} {F}

Parameter

```
G   Roughing axial finishing (E)
Y   Startpoint
Z   Startpoint
Y1= Beginpoint contour
Z1= Beginpoint contour
Y2= Endpoint contour
Z2= Endpoint contour
A   (Angle 1)
B   (Angle 2)
I1= (Chamfer length 1)
R1= (Radius 1)
R2= (Radius 2)
S1= ((Cutting) Speed)
F   (Feed)
```

- Y Starting point. (Recommended value: safety distance up to Y1=)
Position of tool in radial direction. This position is the starting point for finishing.
- Z Starting point. (Recommended value: safety distance up to Z1=)
Position of tool in axial direction. This position is the starting point for finishing.
Finishing starts at Z1= until Z2= is reached.
- Y1= Contour starting point
Starting point of the contour to be machined.
- Z1= Contour starting point
Starting point of the contour to be machined.
- Y2= Contour end point
End point of the contour to be machined.

- Z2= Contour end point
End point of the contour to be machined.
- A Angle: Basic setting A=0.
Angle (>0) at contour starting point. (Z1=)
Angles A and B must be chosen so that the tool does not undercut.
- B Angle: Basic setting B=0.
Angle (>0) at contour end point. (Z2=)
- I1= Chamfer length: Basic setting I1=0.
Chamfer length at start and end of contour.
Only I1= or R1= may be programmed.
- R1= Rounding: Basic setting R1=0.
Rounding at start and end of contour.
- R2= Rounding: Basic setting R2= tool tip radius.
Rounding at the bottom of the contour.
- F Feed rate: Traverse speed of tool in mm/min.
- S1= Spindle speed.

Type of function

non modal

Associated functions

G832 for finishing

Notes and application

Finishing goes from Y1/Z1 to Y1/Z2.

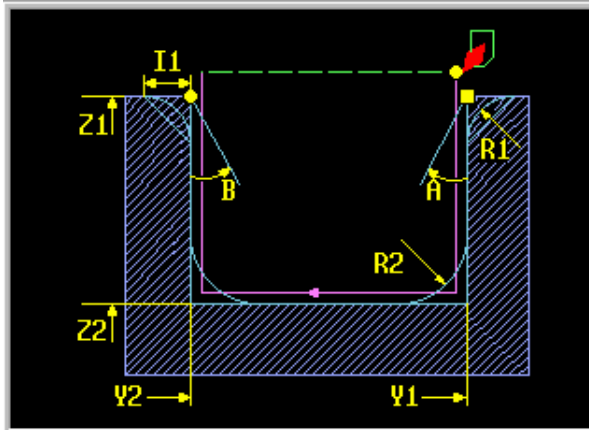
Tool orientation may only be 3, 4 or 5.

The tool path is corrected for the tip radius.

12.10.8 G837 Roughing radial finishing

Purpose

Roughing radial finishing



Format

N... G837 Y.. Z.. Y1=.. Z1=.. Y2=.. Z2=.. {A..} {B..} [{I1=..}]{R1=..} {R2=..} {S1=..} {F}

Parameter

```
G  Roughing radial finishing (E)
Y  Startpoint
Z  Startpoint
Y1= Beginpoint contour
Z1= Beginpoint contour
Y2= Endpoint contour
Z2= Endpoint contour
A  (Angle 1)
B  (Angle 2)
I1= (Chamfer length 1)
R1= (Radius 1)
R2= (Radius 2)
S1= ((Cutting) Speed)
F  (Feed)
```

- Y Starting point. (Recommended value:safety distance up to Y1=)
Position of tool in radial direction. This position is the starting point for finishing.
Finishing starts at Y1= until Y2= is reached.
- Z Starting point. (Recommended value: safety distance up to Z1=)
Position of tool in radial direction. This position is the starting point for finishing.
- Y1= Contour starting point
Starting point of the contour to be machined.
- Z1= Contour starting point
Starting point of the contour to be machined.
- Y2= Contour end point
End point of the contour to be machined.

- Z2= Contour end point.
End point of the contour to be machined.
- A Angle: Basic setting A=0.
Angle (>0) at contour starting point. (Y1=)
Angles A and B must be chosen so that the tool does not undercut.
- B Angle: Basic setting B=0.
Angle (>0) at contour end point. (Y2=)
- I1= Chamfer length: Basic setting I1=0.
Chamfer length at start and end of contour.
Only I1= or R1= may be programmed.
- R1= Rounding: Basic setting R1=0.
Rounding at start and end of contour.
- R2= Rounding: Basic setting R2= tool tip radius.
Rounding at the bottom of the contour.
- F Feed rate: Traverse speed of tool in mm/min.
- S1 Spindle speed.

Type of function

non modal

Associated functions

G833 for finishing

Notes and application

Finishing goes from Y1/Z1 to Y1/Z2.

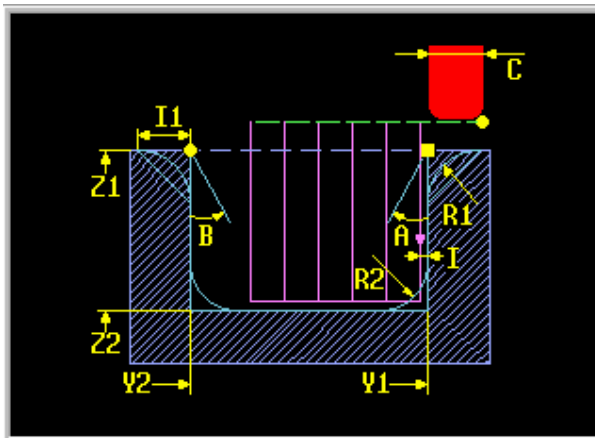
Tool orientation may only be 5, 6 or 7.

The tool path is corrected for the tip radius.

12.10.9 G842 Grooving axial

Purpose

Grooving axial



Format

N... G842 Y.. Z.. Y1=.. Z1=.. Y2=.. Z2=.. C.. {A..} {B..} [{I1=..}]{R1=..} {R2=..} {I..} {S1=..} {F}

Parameter

G Grooving axial (E)
 Y Startpoint
 Z Startpoint
 Y1= Beginpoint contour
 Z1= Beginpoint contour
 Y2= Endpoint contour
 Z2= Endpoint contour
 C Tool width
 A (Angle 1)
 B (Angle 2)
 I1= (Chamfer length 1)
 R1= (Radius 1)
 R2= (Radius 2)
 I (Finishing)

S1= ((Cutting) Speed)
 F (Feed)

- Y Starting point. (Recommended value: safety distance up to Y1=)
 Position of tool in radial direction. This position is the starting point for machining.
 Machining starts at Y1= with the feed width until Y2= is reached.
- Z Starting point. (Recommended value: safety distance up to Z2=)
 Position of tool in axial direction. This position is the starting point for machining.
- Y1= Contour starting point
 Starting point of the contour to be machined.
- Z1= Contour starting point.
 Starting point of the contour to be machined.
- Y2= Contour end point.
 End point of the contour to be machined.

- Z2= Contour end point.
End point of the contour to be machined.
- C Chisel width.
Width of tool. The feed width is C minus twice the tip radius.
- A Angle: Basic setting A=0.
Angle (>0) at contour starting point. (Y1=)
- B Angle: Basic setting B=0.
Angle (>0) at contour end point. (Y2=)
- I1= Chamfer length: Basic setting I1=0.
Chamfer length at start and end of contour.
Only I1= or R1= may be programmed.
- R1= Rounding: Basic setting R1=0.
Rounding at start and end of contour.
- R2= Rounding: Basic setting R2= tool corner radius.
Rounding at the bottom of the contour.
- I Finishing allowance: basic setting I=0.
- F Feed rate: Traverse speed of tool in mm/min.
- S1= Spindle speed.

Type of function

non modal

Associated functions

G846 for finishing

Notes and application

Rough cutting takes place first, then finishing.

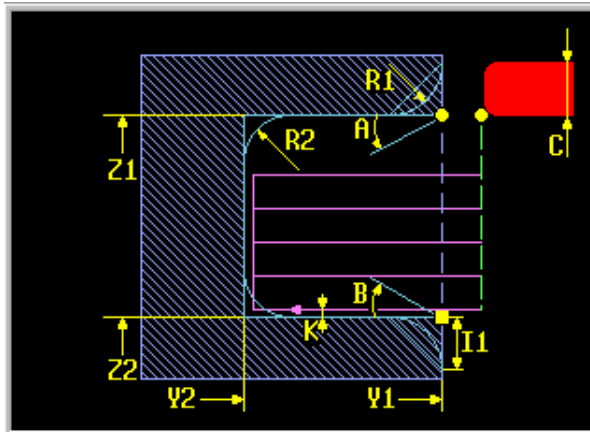
Tool orientation may only be 5, 6 or 7.

The tool path is corrected for the tip radius.

12.10.10 G843 Grooving radial

Purpose

Grooving radial



Format

N... G843 Y.. Z.. Y1=.. Z1=.. Y2=.. Z2=.. C.. {A..} {B..} [{I1=..}]{R1=..} {R2=..} {K..} {S1=..} {F}

Parameter

G Grooving radial (E)
 Y Startpoint
 Z Startpoint
 Y1= Beginpoint contour
 Z1= Beginpoint contour
 Y2= Endpoint contour
 Z2= Endpoint contour
 C Tool width
 A (Angle 1)
 B (Angle 2)
 I1= (Chamfer length 1)
 R1= (Radius 1)
 R2= (Radius 2)
 K (Finishing)

S1= ((Cutting) Speed)
 F (Feed)

- Y Starting point. (Recommended value: safety distance up to Y1=).
 Position of tool in radial direction. This position is the starting point for machining.
 Machining starts at Y until Y2 is reached.
- Z Starting point. (Recommended value: safety distance up to Z2=).
 Position of tool in axial direction. This position is the starting point for machining.
 Machining starts at Z2= with the feed width until Z1= is reached.
- Y1= Contour starting point.
 Starting point of the contour to be machined.
- Z1= Contour starting point.
 Starting point of the contour to be machined.
- Y2= Contour end point.
 End point of the contour to be machined.

- Z2= Contour end point.
End point of the contour to be machined.
- C Chisel width
Width of tool. The feed width is C minus twice the tip radius.
- A Angle: Basic setting A=0.
Angle (>0) at contour starting point (Z1=)
- B Angle: Basic setting B=0.
Angle (>0) at contour end point. (Z2=)
- I1= Chamfer length: Basic setting I1=0.
Chamfer length at start and end of contour.
Only I1= or R1= may be programmed.
- R1= Rounding: Basic setting R1=0.
Rounding at start and end of contour.
- R2= Rounding: Basic setting R2= tool tip radius.
Rounding at the bottom of the contour.
- K Finishing allowance: basic setting K=0.
- F Feed rate: Traverse speed of tool in mm/min.
- S1= Spindle speed.

Type of function

non modal

Associated functions

G847 for finishing

Notes and application

Rough cutting takes place first, then finishing.

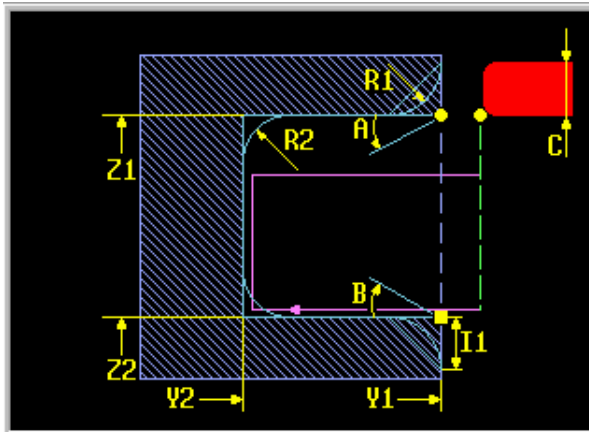
Tool orientation may only be 3, 4 or 5.

The tool path is corrected for the tip radius.

12.10.11 G846 Grooving axial finishing

Purpose

Grooving axial finishing



Format

N... G846 Y.. Z.. Y1=.. Z1=.. Y2=.. Z2=.. C.. {A..} {B..} [{I1=..}] {R1=..} {R2=..} {I..} {S1=..} {F}

Parameter

G Grooving axial finishing (E)
 Y Startpoint
 Z Startpoint
 Y1= Beginpoint contour
 Z1= Beginpoint contour
 Y2= Endpoint contour
 Z2= Endpoint contour
 C Tool width
 A (Angle 1)
 B (Angle 2)
 I1= (Chamfer length 1)
 R1= (Radius 1)
 R2= (Radius 2)
 I (Finishing)

S1= ((Cutting) Speed)
 F (Feed)

- Y Starting point. (Recommended value: safety distance up to Y1=)
 Position of tool in radial direction. This position is the starting point for machining.
 Machining starts at Y until Y2 is reached.
- Z Starting point. (Recommended value: safety distance up to Z1=).
 Position of tool in axial direction. This position is the starting point for machining.
 Starts at Z2= until Z1= is reached.
- Y1= Contour starting point.
 Starting point of the contour to be machined.
- Z1= Contour starting point
 Starting point of the contour to be machined.
- Y2= Contour end point
 End point of the contour to be machined.

- Z2= Contour end point.
End point of the contour to be machined.
- C Chisel width.
Width of tool. The feed width is C minus twice the corner radius.
- A Angle: Basic setting A=0.
Angle (>0) at contour starting point. (Y1=)
- B Angle: Basic setting B=0.
Angle (>0) at contour end point. (Y2=).
- I1= Chamfer length: Basic setting I1=0.
Chamfer length at start and end of contour.
Only I1= or R1= may be programmed.
- R1= Rounding: Basic setting R1=0.
Rounding at start and end of contour.
- R2= Rounding: Basic setting R2= tool tip radius.
Rounding at the bottom of the contour.
- I Finishing allowance: basic setting I=0.
- F Feed rate: Traverse speed of tool in mm/min.
- S1= Spindle speed.

Type of function

non modal

Associated functions

G842 for finishing.

Notes and application

Finishing goes from Y1/Z1 to Y1/Z2.

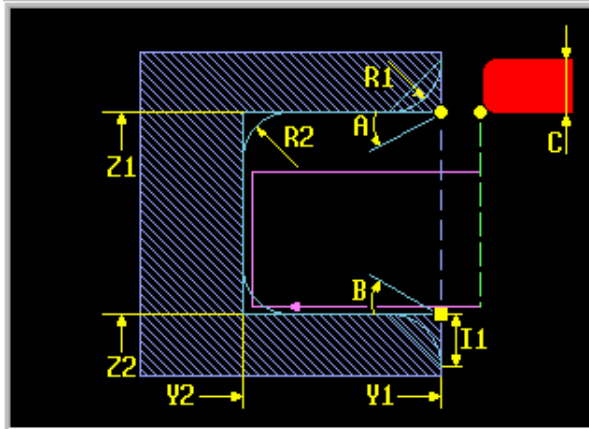
Tool orientation may only be 5, 6 or 7.

The tool path is corrected for the tip radius.

12.10.12 G847 Grooving radial finishing

Purpose

Grooving radial finishing



Format

N... G847 Y.. Z.. Y1=.. Z1=.. Y2=.. Z2=.. C.. {A..} {B..} [{I1=..}]{R1=..} {R2=..} {K..} {S1=..} {F}

Parameter

G Grooving radial finishing (E)
 Y Startpoint
 Z Startpoint
 Y1= Beginpoint contour
 Z1= Beginpoint contour
 Y2= Endpoint contour
 Z2= Endpoint contour
 C Tool width
 A (Angle 1)
 B (Angle 2)
 I1= (Chamfer length 1)
 R1= (Radius 1)
 R2= (Radius 2)
 K (Finishing)

S1= ((Cutting) Speed)
 F (Feed)

- Y Starting point. (Recommended value: safety distance up to Y1=).
 Position of tool in radial direction. This position is the starting point for finishing.
 Finishing starts at Y until Y2 is reached.
- Z Starting point. (Recommended value: safety distance up to Z2=).
 Position of tool in axial direction. This position is the starting point for finishing.
- Y1= Contour starting point.
 Starting point of the contour to be machined.
- Z1= Contour starting point.
 Starting point of the contour to be machined.
- Y2= Contour end point.
 End point of the contour to be machined.
- Z2= Contour end point.

End point of the contour to be machined.

- C Chisel width.
 Width of tool. The feed width is C minus twice the corner radius.
- A Angle: Basic setting A=0.
 Angle (>0) at contour starting point. (Z1=).
- B Angle: Basic setting B=0.
 Angle (>0) at contour end point. (Z2=).
- I1= Chamfer length: Basic setting I1=0.
 Chamfer length at start and end of contour.
 Only I1= or R1= may be programmed.
- R1= Rounding: Basic setting R1=0.
 Rounding at start and end of contour.
- R2= Rounding: Basic setting R2= tool tip radius.
 Rounding at the bottom of the contour.
- K Finishing allowance: basic setting K=0.
- F Feed rate: Traverse speed of tool in mm/min.
- S1= Spindle speed.

Type of function

non modal

Associated functions

G843 for roughing.

Notes and application

Finishing goes from Y1/Z2 to Y1/Z1.

Tool orientation may only be 3, 4 or 5.

The tool path is corrected for the tip radius.

12.11 Examples

Example 1:

```

N9999
N1 G17
N2 G37
N3 M54
N4 T1 M6
N5 S1000 F1000 M3
N...

N100 G17 Z1=1 Y1=2
N101 G36
N102 T7 M6
N103 S1=100 M1=3
N104 G0 X0 Y100 Z100
N105 G822 ....
N...

N200 G17
N201 G37
N203 T1 M6
N204 S1000 M3
N205 ....
N300 M30

```

Explanation:

```

N1      Set planes for milling.
        Length compensation in Z direction.
N2      Milling mode
N3      Head is in the Z direction
N4      Insert milling tool
N5      Start Spindle
N...
N...    Milling

N100    Set planes for turning.
        Main axis 1 is Z, main axis 2 is Y.
        Radius correction in ZY plane.
N101    Turning
N102    Insert tool
N103    Start rotary table for continuous turning.
N104    Position turning tool.
N105    Start clearance axial cycles.
N...
N...    Turning

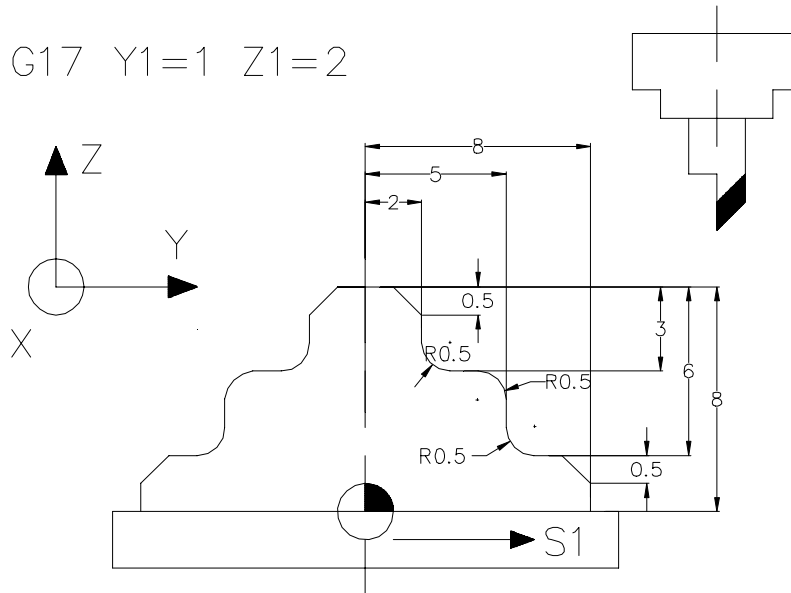
N200    Set planes for milling.
        Length compensation in Z direction.
N201    Milling mode
N202    Insert milling tool
N103    Start Spindle
N104    Position milling tool
N...
N...    Milling

N300

```

Example 2:

N9999
N1 G17
N2 G37
N3 G54 I1 Z8
N4 G36 (continuous turning)
N5 M54
N6 G17 Z1=1 Y1=2
N7 G195 X-1 Y-1 Z1 I2 J12 K-11
N8 G199 X0 Y0 Z0 B4 C2
N9 G198 I1=14 X0 Y8 Z0
N10 G2 X0 Y8 I0 J0
N11 G1 X0 Y8 Z-8
N12 G2 X0 Y8 I0 J0
N13 G196
N14 T1 M6 (L100 R5 C0.3 Q3=800)
N15 S1=1000 M1=3
N16 G0 X0 Y8 Z3 F1000 S1000 M5
N17 (roughing)
N18 G823 Y8 Z0.3 Y1=8 Z1=-3 Y2=2 Z2=0 I1=0.5 R2=0.5 C0.2
N19 G823 Y8 Z-2.7 Y1=8 Z1=-6 Y2=5 Z2=-3 R1=0.5 I2=0.5 R2=0.5 C0.2
N20 (roughing)
N21 G827 Y8 Z-6.7 Y1=8 Z1=-6 Y2=5 Z2=-3 R1=0.5 I2=0.5 R2=0.5
N22 G827 Y8 Z-2.7 Y1=8 Z1=-3 Y2=2 Z2=0 I1=0.5 R2=0.5
N23 G0 Z10
N24 T0 M6
N25 G37
N26 G53
N27 M30



Explanation:

- | | |
|----|--|
| N1 | Set planes for milling. Length compensation in Z direction. |
| N2 | Milling mode |
| N3 | Zero point displacement for Z direction. Upper edge of material is zero. |
| N4 | Turning |
| N5 | Head is in the Z direction. |
| N6 | Set planes for turning. |

Main axis 1 is Z, main axis 2 is Y.
 Radius correction in ZY plane.

N7 Set graphics window.
 N8 Start of material graphical contour description. B4 means automatic drawing.
 N9 Start of contour description. I1=14 is light blue colour.
 N10 Upper circle of cylinder.
 N11 Line
 N12 Lower circle of cylinder.
 N13 End of graphical contour description.
 N14 Insert turning tool (length, radius, corner radius and type).
 N15 Start rotary table for continuous turning.
 N16 Position turning tool
 N18 G823 start clearance radial cycles. Turn upper part
 Y8 Starting point
 Z0.3 Starting point
 Y1=8 Contour starting point
 Z1=-3 Contour starting point
 Y2=2 Contour end point
 Z2=0 Contour end point
 C0.2 Feed depth
 I1=0.5 Chamfer length 1
 R2=0.5 Radius 2
 N19 G823 start clearance radial cycles. Turn lower part
 Y8 Starting point
 Z-2.7 Starting point
 Y1=8 Contour starting point
 Z1=-6 Contour starting point
 Y2=5 Contour end point
 Z2=0 Contour end point
 C0.2 Feed depth
 R1=0.5 Radius 1
 I2=0.5 Chamfer length 2
 R2=0.5 Radius 2
 N21 G827 start clearance radial finishing cycles. Finish machine lower part.
 Y8 Starting point
 Z-6.7 Starting point
 Y1=8 Contour starting point
 Z1=-6 Contour starting point
 Y2=5 Contour end point
 Z2=-3 Contour end point
 R1=0.5 Radius 1
 I2=0.5 Chamfer length 1
 R2=0.5 Radius 2
 N22 G827 start clearance radial finishing cycles. Finish machine upper part
 Y8 Starting point
 Z-2.7 Starting point
 Y1=8 Contour starting point
 Z1=-3 Contour starting point
 Y2=2 Contour end point
 Z2=0 Contour end point
 I1=0.5 Chamfer length 1
 R2=0.5 Radius 2
 N23 Move tool clear
 N24 Reset tool
 N25 Milling mode
 N26 Deactivate zero point displacement
 N27 Program end

13. G64 Geometric calculations with continuous movements

CONVENTIONS WITH THE FORMATS

For all the formats in this appendix the G64-function is assumed to have already been programmed in a previous block and is therefore active.

The XY-plane is also assumed to be the active plane; if another plane is active the appropriate addresses must be substituted in the formats.

To show that more than one block is required with a particular format, the first block is numbered as N1 and the following as N2, N3 etc. The use of these numbers is not compulsory; they have been used purely as a convention.

In the formats, programming the end point is indicated with X.. Y... but instead of these coordinates the polar coordinates B2=, L2=.. or a defined point P.. can be used too.

Sometimes, programming the endpoint is indicated with [endpoint]. In this case the endpoint can be programmed as outside the geometry (G63 active). Thus with:
X.. only, or Y.. only, or X.. and Y.. or B1=.. and X.. or B1=.. and Y..

In the formats, programming the centre point of a circle is indicated with I.. J... but instead of these coordinates the polar coordinates B3=, L3=.. can be used.

The use of a support point is indicated with X.. Y.. I1=0 and of a parallel line with X.. Y.. I1= ... Instead of X.. and Y.. the polar coordinates B2=, L2=.. or a defined point P.. can be used too. It is also possible to use X1=.. Y1=.. Instead of X.. Y.. I1=0 In some cases a support point or a parallel line can be used. This is indicated with X.. Y.. I1=0 or I1=+...

Sometimes a support point is indicated with [support point]. All possible formats for support point can be used.

In the illustrations in which the formats are explained, the following conventions are used:

P_o = a start point known from the previous block
P_s = a support point on a line or on a parallel line
P_e = a programmed end point
M = a programmed circle centre point
R = a programmed radius of a circle

A lot of line definitions are given with an angle B1=.. or a support point P, on the line. This is indicated in the illustrations with {B1=} and {Ps}. If B1= and Ps are drawn without the () both words have to be programmed.

Contents of this section

Intersection point

- 1.1 Intersection point of two straight lines
- 1.2 Intersection point of two lines programmed as end point
- 1.3 Chamfer between intersecting straight lines
- 1.4 Rounding between intersecting straight lines
- 1.5 Rounding between straight line and chamfer
- 1.6 Intersection point indicator (j1=)
- 1.6 Intersection point between line and circle
- 1.7 Intersection point of line and circle programmed as end point
- 1.8 Rounding between intersecting line and circle
- 1.9 Intersection point between circle and line
- 1.10 Intersection point of circle and line programmed as end point
- 1.11 Rounding between intersecting circle and line
- 1.12 Intersection point between two circles
- 1.13 Intersection point of two circles programmed as end point
- 1.14 Rounding between two intersecting circles

Point of tangency

- 2.1 Point of tangency indicator (r1=)
- 2.2 Tangent line and circle
- 2.3 Connecting circle between tangent line and circle
- 2.4 Tangent circle and line
- 2.5 Connecting circle between tangent circle and line
- 2.6 Two tangent circles
- 2.7 Continuous connecting circle between two tangent circles

Connecting circle between elements, which do not meet

- 3.1 Line and circle
- 3.2 Circle and line
- 3.3 Two circles outside each other
- 3.4 One circle inside the other one
- 3.5 Concentric circles

Geometric calculations with non-continuous movements G64

- 4.1 Rounding or connecting circle indicator (k1=)
- 4.2 Rounding with intersection points
- 4.3 Rounding between intersecting straight lines
- 4.4 Rounding between intersecting line and circle
- 4.5 Rounding between intersecting circle and line
- 4.6 Rounding between two intersecting circles
- 4.7 Tangent lines (r1=)
- 4.8 Connecting circle between tangent line and circle or v.v.
- 4.9 Connecting circle between a line which do not meet a circle
- 4.10 Connecting circle between circles outside each other
- 4.11 Connecting circle between two circles one inside the other
- 4.12 Connecting circle with two concentric circles

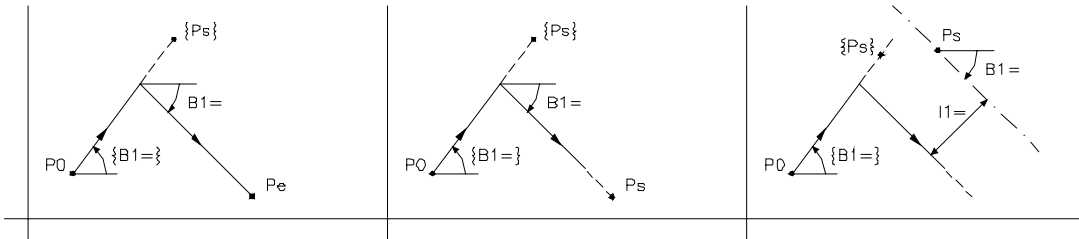
13.1 Intersection point

13.1.1 Intersection point of two straight lines

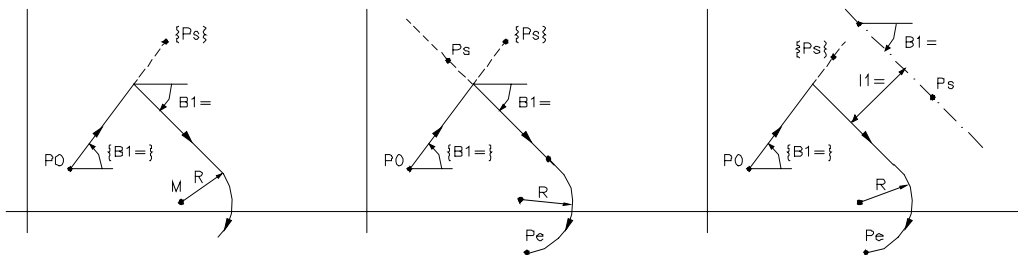
To calculate the intersection point between two lines

Start point from N1 is known

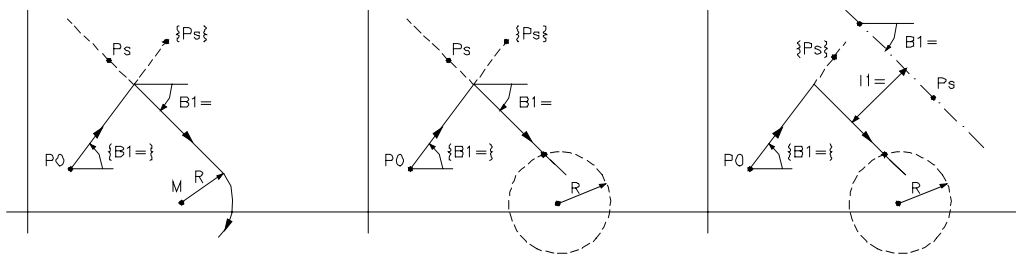
If the start point of the first line is known either the angle with the main axis or any support point on the line can be used to define the first line. Several formats are possible for the second line.



N1	G1	B1=..			
N2		B1=..	X..	Y..	
or					
N1	G1	B1=..			
N2		B1=..	X..	Y..	I1=0 or I1=±..
or					
N1	G1	X..	Y..	I1=0	
N2		B1=..	X..	Y..	
or					
N1	G1	X..	Y..	I1=0	
N2		B1=..	X..	Y..	I1=0 or I1=±..



N1	G1	B1=..			
N2		B1=..	R1=0		
or					
N1	G1	B1=..			
N2		B1=..	X..	Y..	I1=0 or I1=±.. R1=0
or					
N1	G1	X..	Y..	I1=0	
N2		B1=..	R1=0		
or					
N1	G1	X..	Y..	I1=0	
N2		B1=..	X..	Y..	I1=0 R1=0



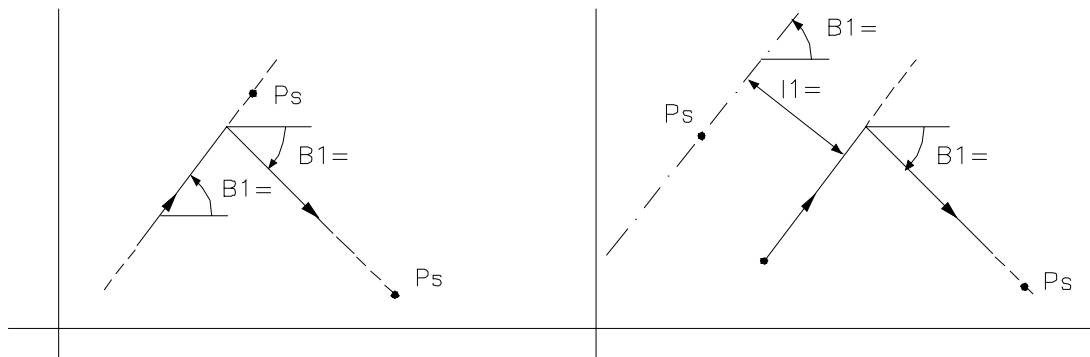
```

N1      G1      B1=..
N2      X..      Y..      I1=0      R1=0
or
N1      G1      B1=..
N2      B1=..    X..      Y..      I1=0 or I1=..      J1=1/2
or
N1      G1      X..      Y..      I1=0
N2      X..      Y..      I1=0      R1=0
or
N1      G1      X..      Y..      I1=0
N2      B1=..    X..      Y..      I1=0 or I1=..      J1=1/2

```

Start point from N1 is not known

If the start point from N1 is not known, both the angle and a support point have to be programmed in block N1. So this block reads:



```

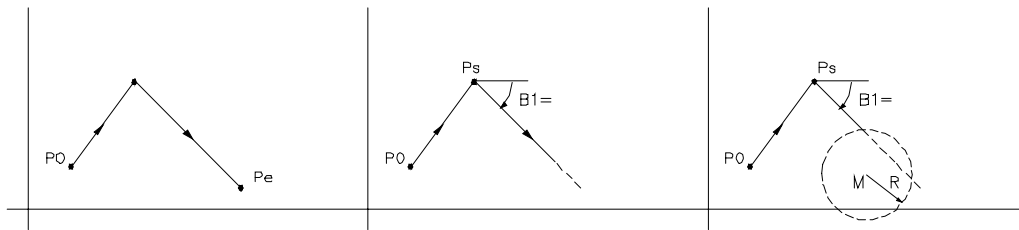
N1      G1      B1=..    X..      Y..      I1=0 or I1=..

```

Block N2 from the mentioned cases remains the same.

support point coincides with the point of intersection

If the support point coincides with the point of intersection, it is assumed that this point is the start point of the next line. This results in a few additional formats in which the second line can be programmed with either the angle or a support point on the line:

Start point from N1 is known

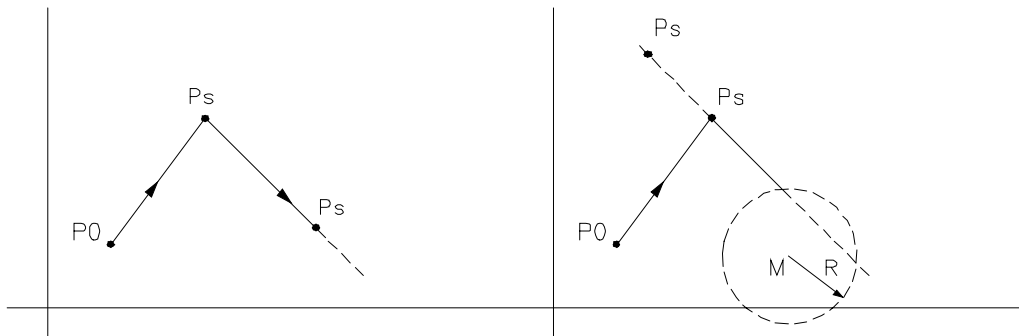
N1 G1 [support point = Intersection point]
 N2 X.. Y..

or

N1 G1 [support point = Intersection point]
 N2 B1=..

or

N1 G1 [support point = Intersection point]
 N2 B1=.. J1=1/2



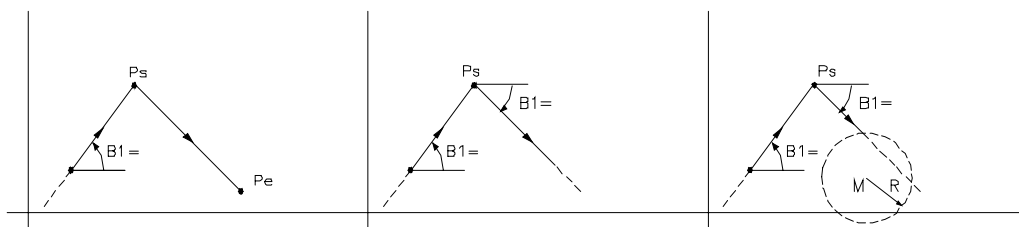
N1 G1 [support point = Intersection point]
 N2 X.. Y.. I1=0

or

N1 G1 [support point = Intersection point]
 N2 X.. Y.. I1=0 J1=1/2

Start point from N1 is not known

If the start point from N1 is not known, both the angle and the support point have to be programmed in block N1. So this block reads::



N1 G1 B1=.. [support point = Intersection point]

Block N2 from the mentioned cases remains the same.

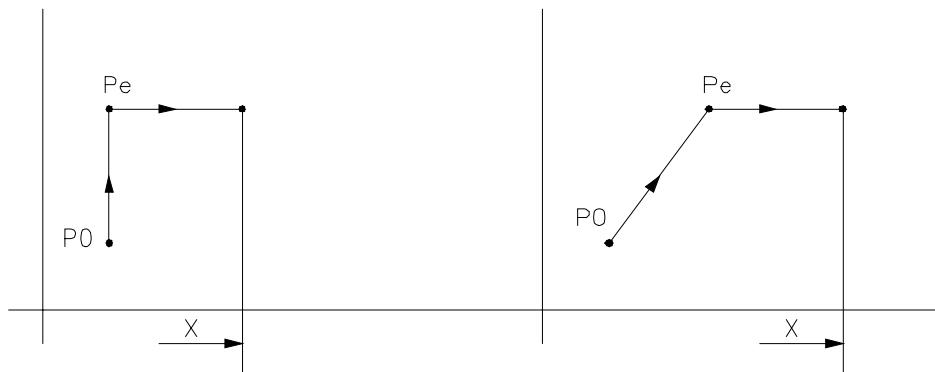
13.1.2 Intersection point programmed as end point

In some cases the intersection point of two lines is known from the drawing and can be programmed directly. It is assumed that this point is the start point of the next line. If the end point is programmed with one coordinate only, the other coordinate is picked up from the previous blocks. The following extra formats are possible:

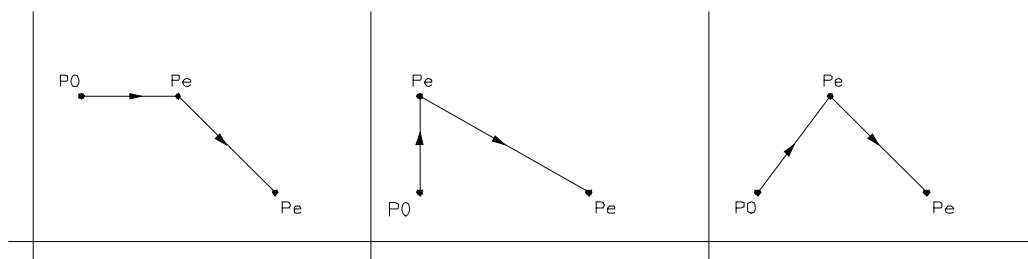
Start point from N1 is known



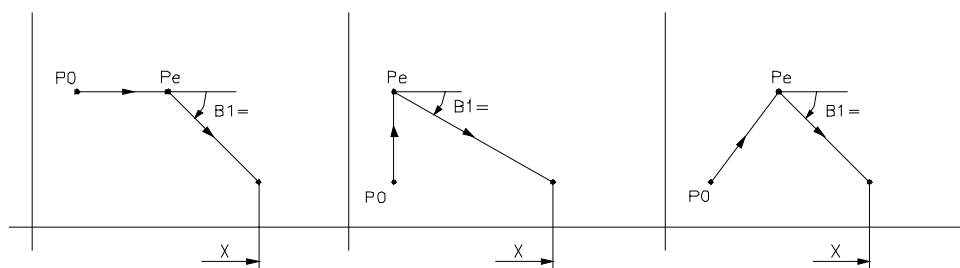
N1	G1	X.. or X..	Y..
N2		Y..	



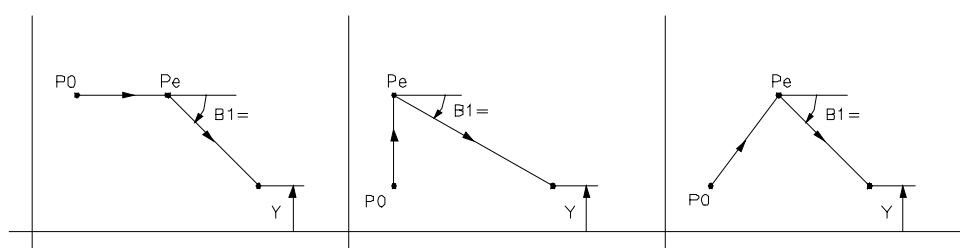
N1	G1	Y.. or X..	Y..
N2		X..	



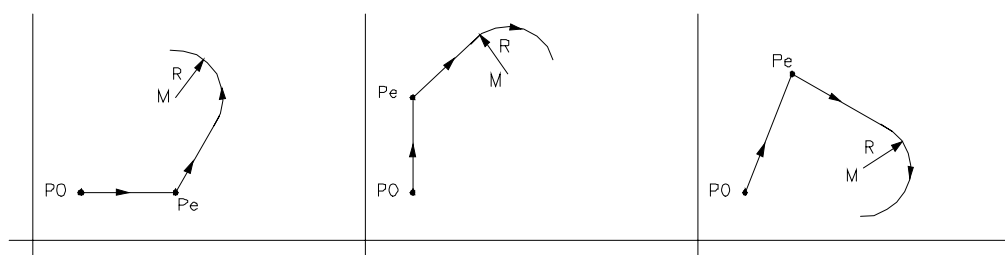
N1	G1	X.. or Y.. or X..	Y..
N2		X.. Y..	



N1 G1 X.. or Y.. or X.. Y..
N2 B1=.. X..



N1 G1 X.. or Y.. or X.. Y..
N2 B1=.. Y..

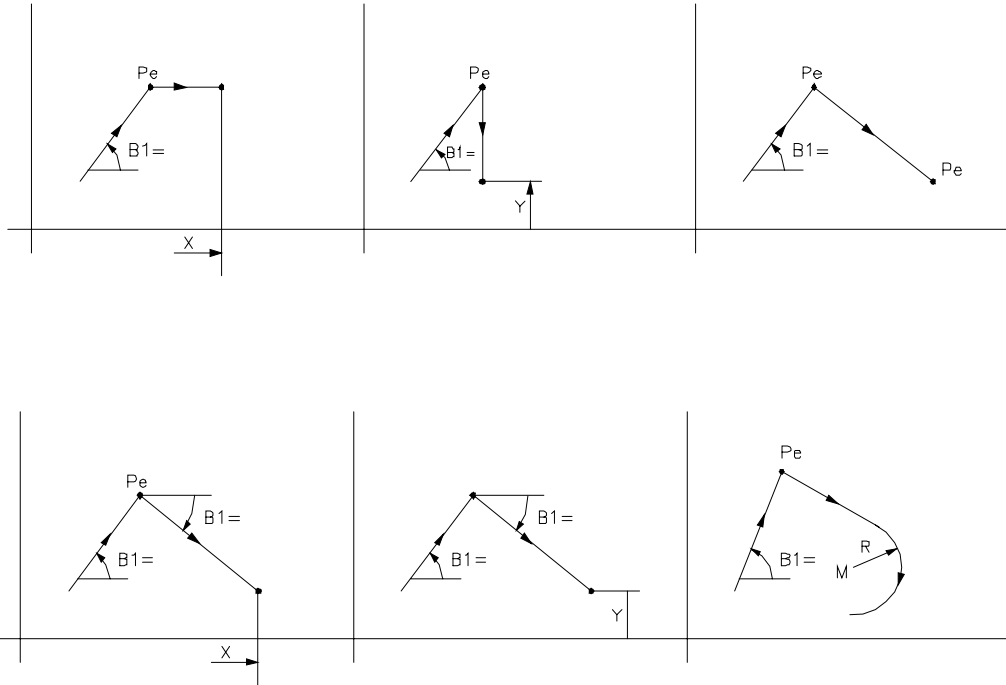


N1 G1 X.. or Y.. or X.. Y..
N2 R1=0

All formats from the previous case for calculating the intersection point can be programmed too.

Start point from N1 is not known

If the start point from N1 is not known, both the angle and the endpoint have to be programmed in block N1. So this block reads:



```

N1    G1    B1=.. X..
or
N1    G1    B1=.. Y..
or
N1    G1    B1=.. X.. Y..

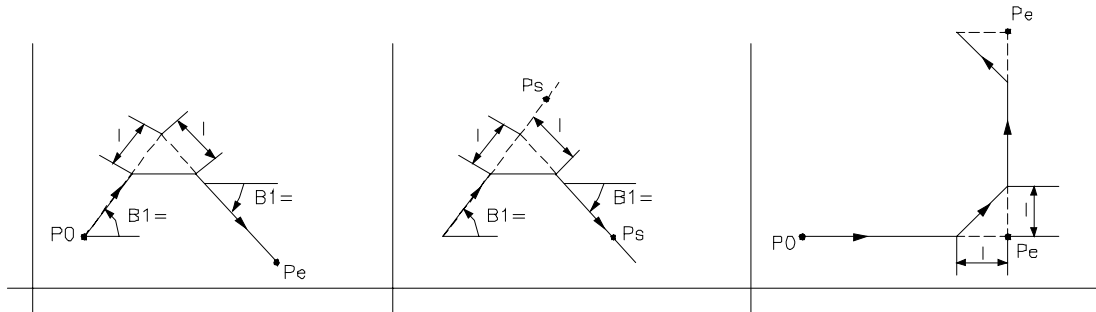
```

Block N2 from the mentioned cases remains the same.

13.1.3 Chamfer between intersecting straight lines

To insert a symmetrical chamfer between two straight lines

Start point from N1 is known



```
N1 G1    B1=..
N2       l...
N3       etc.
```

Refer to block N2 of the previous sections for the formats of block N3.

Note: instead of programming B1=.. in block N1 it is also possible to use a support point, a parallel line or an end point with either X.. or Y.. or X.. and Y..

start point from N1 is not known

If the start point from N1 is not known, both the angle and a support point or end point have to be programmed in block N1. So this block reads:

```
N1 G1 B1=.. X.. Y.. l1=0 or l1=..
N2    l...
N3    etc.
```

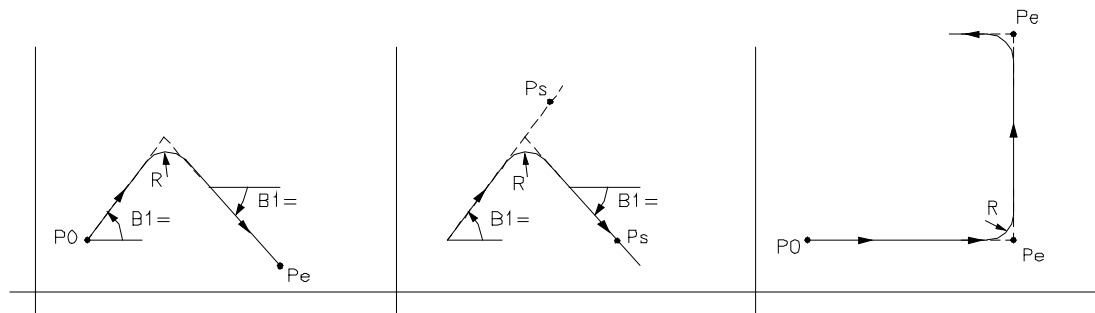
The blocks N2 and N3 are the same as with a known start point. So refer to that section for these blocks.

Note: instead of programming B1=.. with a support point or parallel line in block N1 it is also possible to use an end point with either X.. or K. or X.. and Y..

13.1.4 Rounding between intersecting straight lines

To insert a rounding between two straight lines

Start point from N1 is known



N1	G1	B1=..
N2	G2/G3	R...
N3	G1	etc.

Note: instead of programming $B1=..$ in block N1 it is also possible to use a support point, a parallel line or an end point with either $X..$ or $Y..$ or $X..$ and $Y..$

Refer to block N2 of the previous sections for the formats of block N3.

start point from N1 is not known

If the start point from N1 is not known, both the angle and a support point or end point have to be programmed in block N1. So this block reads:

N1	G1 B1=.. X.. Y.. I1=0 or I1=∇..
N2	G2/G3 R...
N3	G1 etc.

Note: instead of programming $B1=..$ with a support point or parallel line in block N1 it is also possible to use an end point with either $X..$ or $Y..$ or $X..$ and $Y..$

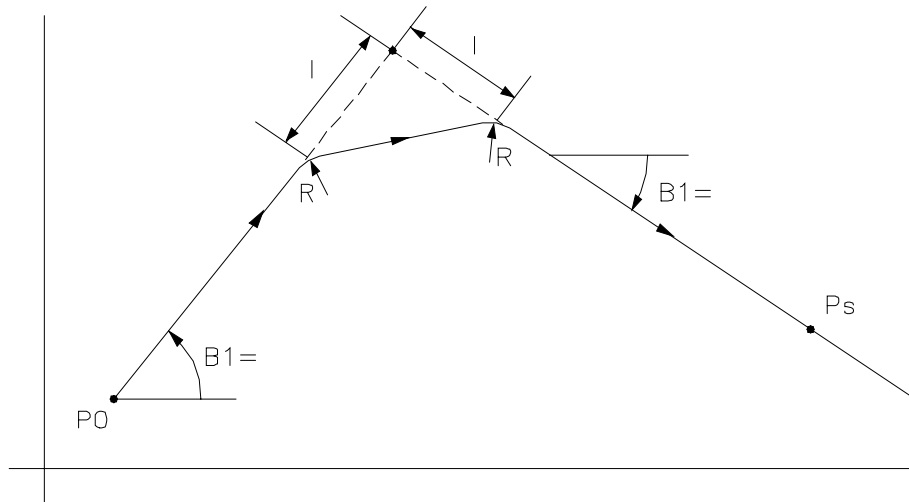
The blocks N2 and N3 are the same as with a known start point. So refer to that section for these blocks.

13.1.5 Rounding between straight line and chamfer

To insert a rounding between a straight line and a chamfer

In the following formats both roundings are indicated. It is possible to insert just one rounding and leave out the other one.

Start point from N1 is known



N1	G1	B1=..
N2	G2/G3	R...
N3	G1	I...
N4	G2/G3	R...
N5	G1	etc.

Note: instead of programming B1=.. in block N1 it is also possible to use a support point, a parallel line or an end point with either X.. or Y.. or X.. and Y..

Refer to block N2 of the previous sections for the formats of block N5.

start point from N1 is not known

If the start point from N1 is not known, both the angle and a support point or end point have to be programmed in block N1. So this block reads:

```

N1 G1 B1=.. X.. Y.. I1=0 or I1=∇..
N2 G2/G3 R...
N3 G1 I...
N4 G2/G3 R...
N5 G1 etc.
```

Note: instead of programming B1=.. with a support point or parallel line in block N1 it is also possible to use an end point with either X.. or Y.. or X.. and Y..

Refer to block N2 of the previous sections for the formats of block N5.

13.1.6 Intersecting point between line circle

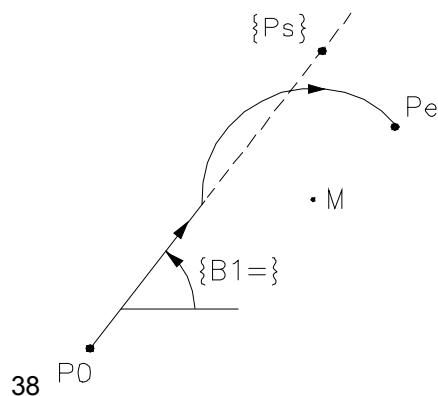
To calculate the point of intersection between line and circle

INTERSECTING POINT INDICATOR (J1=)

Refer to the description of the INTERSECTION POINT INDICATOR in the Notes and Usage of the function G64.

Start point from N1 is known

If the start point of the line is known either the angle the line makes with the main axis or any support point on the line can be used to define the line. Several formats are possible for the circle:

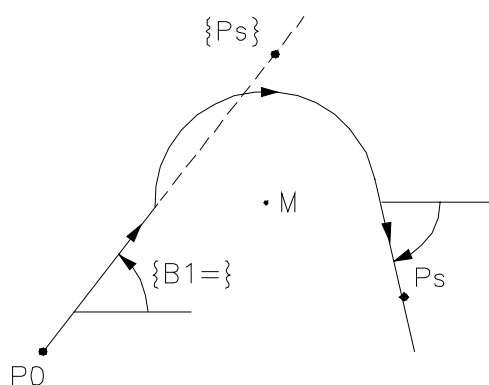
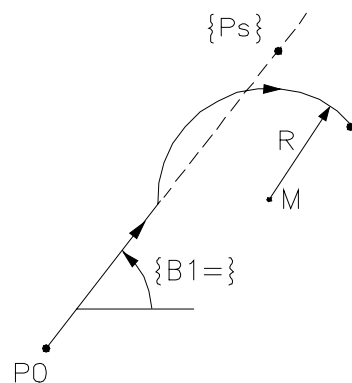


N1	G1	B1=..	J1=1/2		
N2	G2/G3	I..	J..	X..	Y..

or					
N1	G1	B1=..	J1=1/2		
N2	G2/G3	I..	J..	R..	

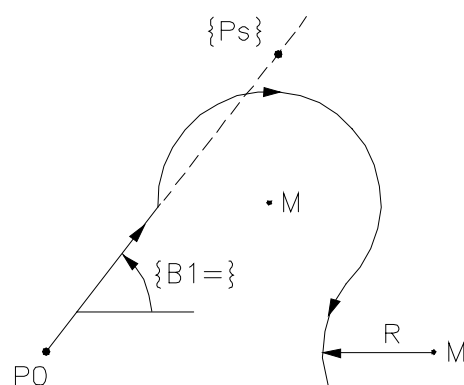
or					
N1	G1	X..	Y..	I1=0	J1=1/2
N2	G2/G3	I..	J..	X..	Y..

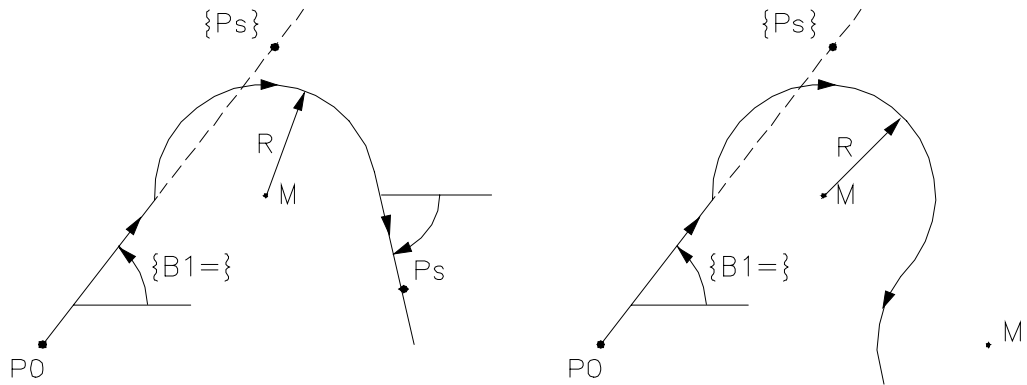
or					
N1	G1	X..	Y..	I1=0	J1=1/2
N2	G2/G3	I..	J..	R..	



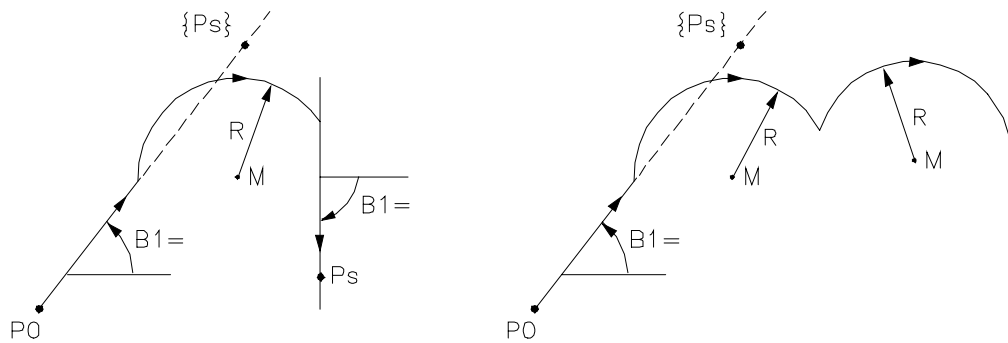
N1	G1	B1=..	J1=1/2		
N2	G2/G3	I..	J..	R1=0	

or					
N1	G1	X..	Y..	I1=0	J1=1/2
N2	G2/G3	I..	J..	R1=0	





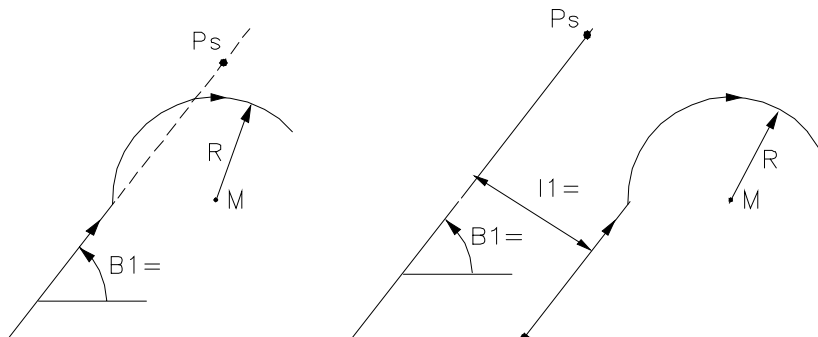
N1	G1	B1=..	J1=1/2		
N2	G2/G3	I..	J..	R..	R1=0
or					
N1	G1	X..	Y..	I1=0	J1=1/2
N2	G2/G3	I..	J..	R..	R1=0



N1	G1	B1=..	J1=1/2		
N2	G2/G3	I..	J..	R..	J1=1/2
or					
N1	G1	X..	Y..	I1=0	J1=1/2
N2	G2/G3	I..	J..	R..	J1=1/2

Start point from N1 is not known

If the start point from N1 is not known, both the angle and a support point have to be programmed in block N1. So this block reads:



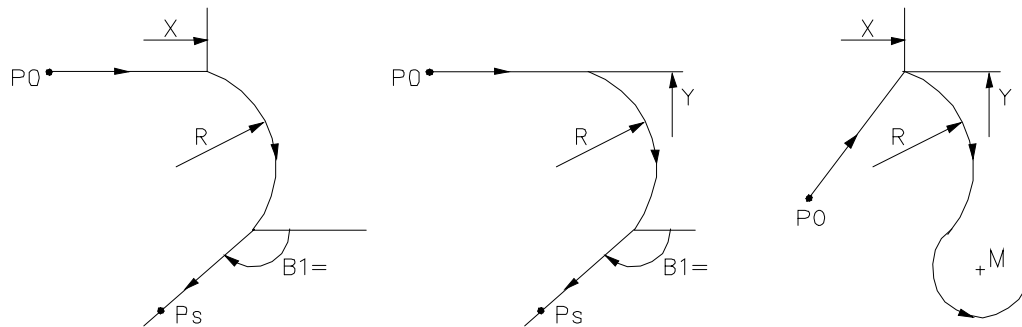
N1	G1	B1=..	X..	Y..	I1=0 or I1=..	J1=1/2
----	----	-------	-----	-----	---------------	--------

Block N2 from the mentioned cases remains the same.

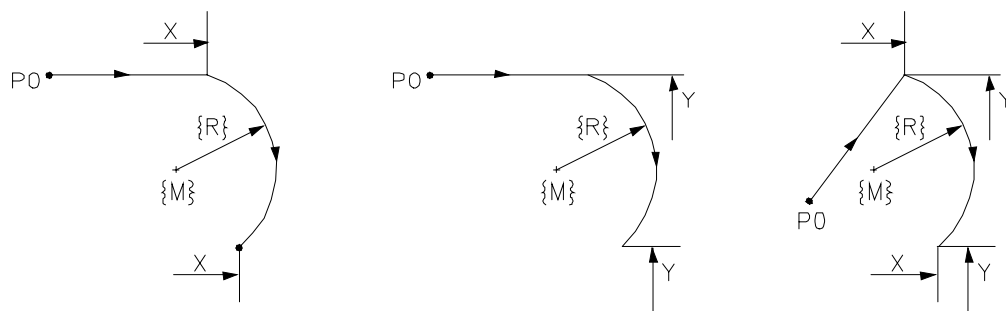
13.1.7 Intersecting point of line and circle programmed as end point

In some cases the intersection point of the line and circle is known from the drawing and can be programmed directly. It is assumed that this point is the start point of the next movement. The end point can be programmed with one or two coordinates and if the start point of the line is not known, the angle, which the line makes with the main axis, can be added to the block. The following formats are possible:

Start point from N1 is known

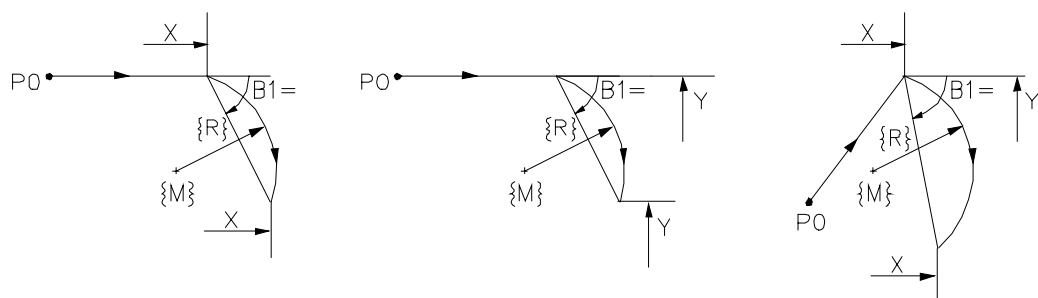


N1	G1	X.. or	Y.. or	X..	Y..
N2	G2/G3	R..	R1=0		



N1	G1	X.. or	Y.. or	X..	Y..
N2	G2/G3	R..		X.. or	Y.. or

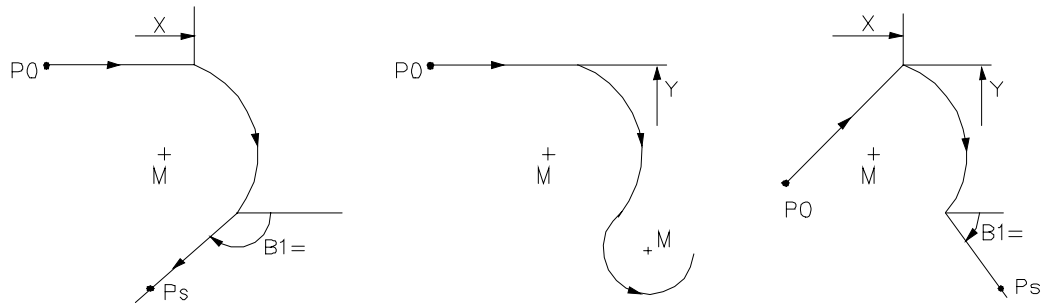
or					
N1	G1	X.. or	Y.. or	X..	Y..
N2	G2/G3	I..		J..	X.. or



N1	G1	X.. or	Y.. or	X..	Y..
N2	G2/G3	R..		B1=..	X.. or

or					
N1	G1	X.. or	Y.. or	X..	Y..
N2	G2/G3	I..		J..	B1=..

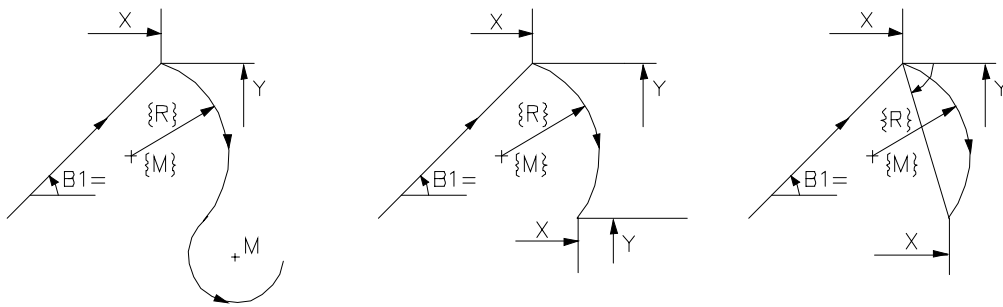
X.. or Y..



N1	G1		X.. or	Y.. or	X..	Y..
N2	G2/G3	I..	J..	{R1=0}	{J1=1/2}	

Start point from N1 is not known

If the start point from N1 is not known, the angle, which the line makes with the main axis, has to be programmed in block N1 too. So this block reads:



N1	G1	B1=..	X..	
or				
N1	G1	B1=..	Y..	
or				
N1	G1	B1=..	X..	Y..

Block N2 from the mentioned cases remains the same.

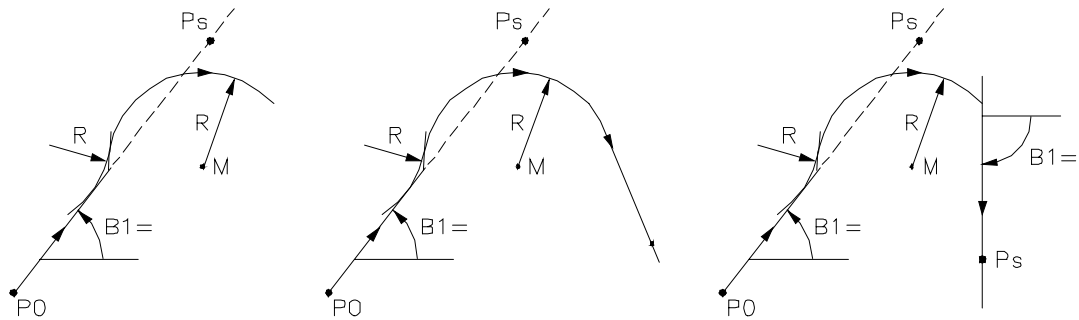
13.1.8 Rounding between intersecting line and circle

To insert a rounding between an intersecting line and a circle

Notice that the direction of rotation of the rounding is opposite to that of the programmed circle.

calculated intersection point

Start point from N1 is known

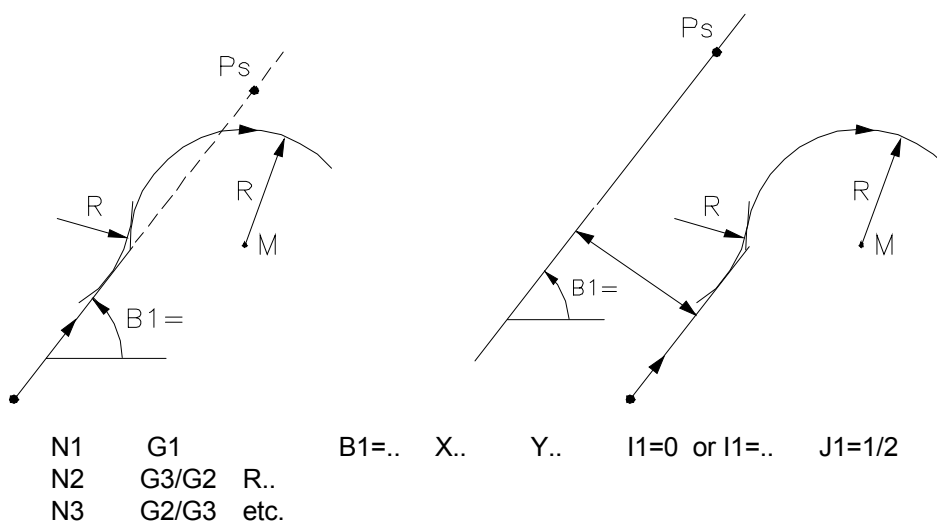


N1	G1	B1=.	J1=1/2
N2	G3/G2	R..	
N3	G2/G3	etc.	
or			
N1	G1	X..	Y.. I1=0 J1=1/2
N2	G3/G2	R..	
N3	G2/G3	etc.	

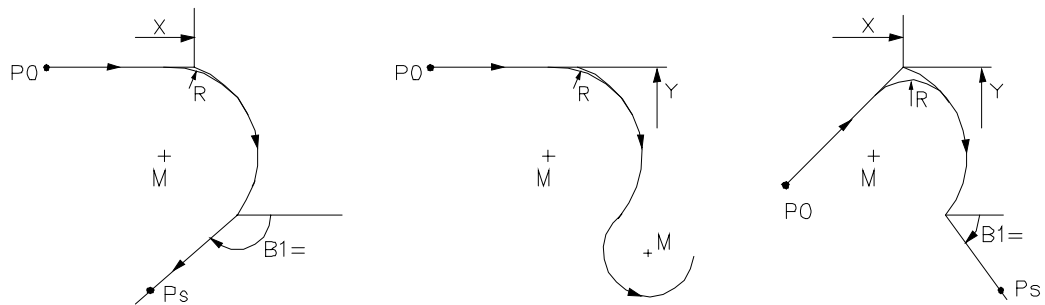
Refer to the section for calculating the intersection point for the formats of block N3.

Start point from N1 is not known

If the start point from N1 is not known, both the angle and a support point have to be programmed in block N1, So this block reads:



Refer to the section with the known start point for the formats of block N3.

Programmed intersecting point**Start point from N1 is known**

N1	G1		X.. or Y.. or X..	Y..
N2	G3/G2	R..		
N3	G2/G3	etc.		

Refer to the section for programming the intersection point for the formats of block N3.

Start point from N1 is not known

If the start point from N1 is not known, the angle, which the line makes with the main axis, has to be programmed in block N1 too. So this block reads:

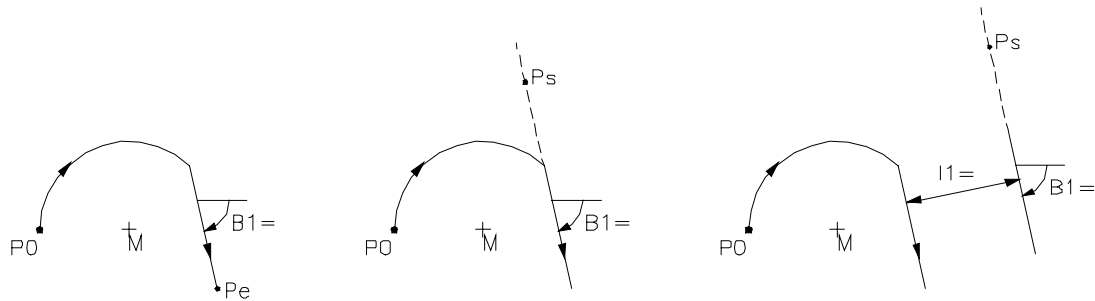
N1	G1		B1=..	X.. or Y.. or X..	Y..
N2	G3/G2	R..			
N3	G2/G3	etc.			

Refer to the section with the known start point for the formats of block N3..

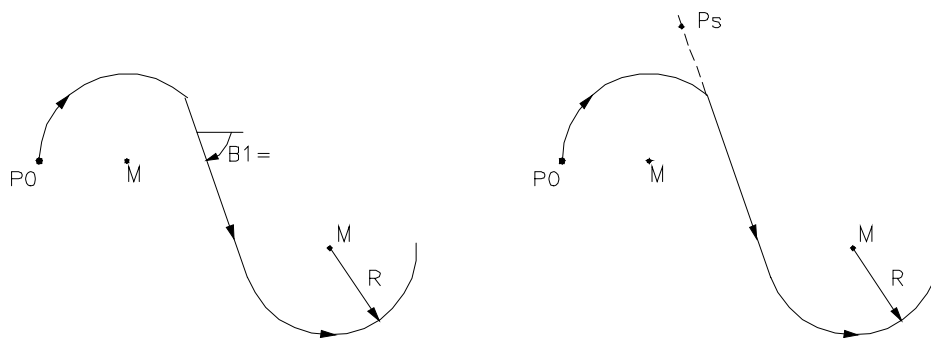
13.1.9 Intersecting point between circle and line

To calculate the intersection point between circle and line

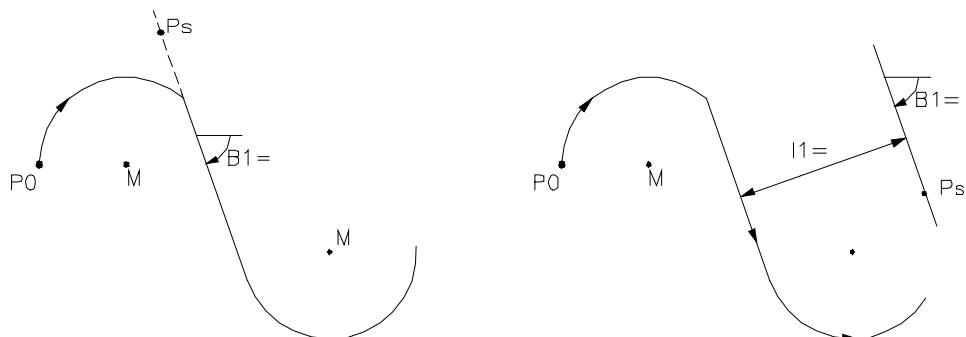
Start point from N1 is known



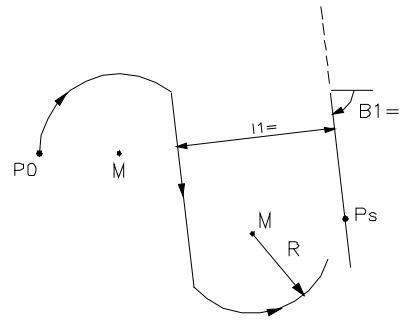
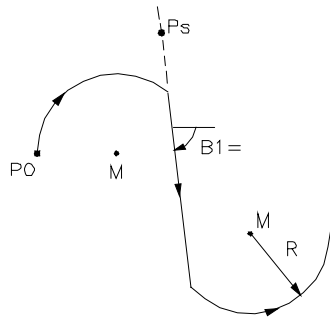
N1	G2/G3	I..	J..	J1=1/2
N2	G1	B1=.. X..	Y..	
or				
N1	G2/G3	I..	J..	J1=1/2
N2	G1	B1=.. X..	Y..	I1=0 or I1=..



N1	G2/G3	I..	J..	J1=1/2
N2	G1	B1=.. R1=0		
or				
N1	G2/G3	I..	J..	J1=1/2
N2	G1	X..	Y..	I1=0 R1=0



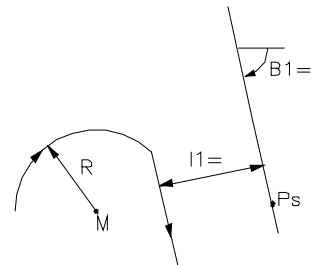
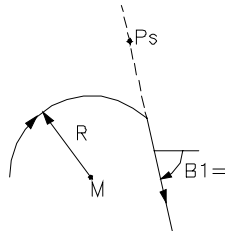
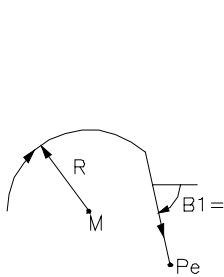
N1	G2/G3	I..	J..	J1=1/2
N2	G1	B1=.. X..	Y..	I1=0 or I1=.. R1=0



N1	G2/G3	I..	J..	J1=1/2
N2	G1	B1=..	X..	Y..
				I1=0 or I1=..
				J1=1/2

Start point from N1 is not known

If the start point is not known, the radius value (R-word) should also be programmed in the block N1, thus this block reads:



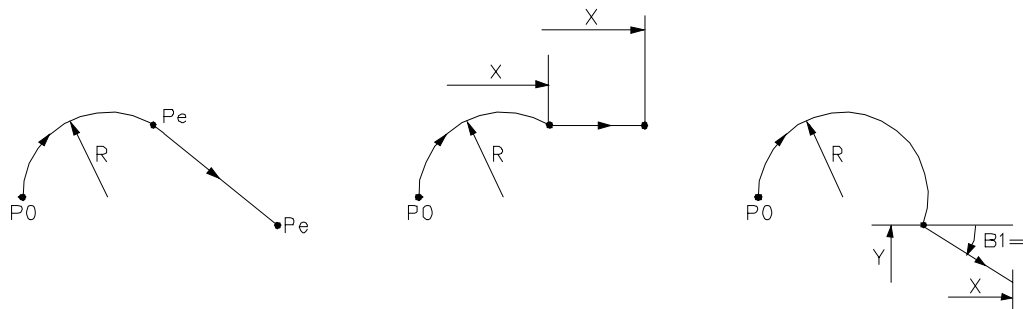
N1	G2/G3	I..	J..	R..	J1=1/2
----	-------	-----	-----	-----	--------

Block N2 from the mentioned cases remains the same.

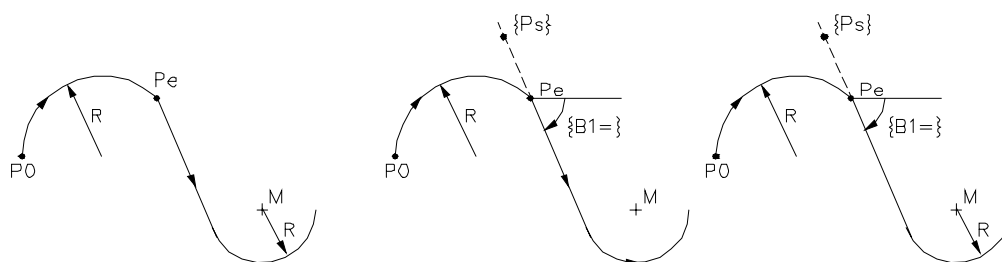
13.1.10 Intersecting point of circle and line programmed as end point

In some cases the intersection point of the circle and line is known from the drawing and can be programmed directly. It is assumed that this point is the start point of the next movement. The following formats are possible:

Start point from N1 is known



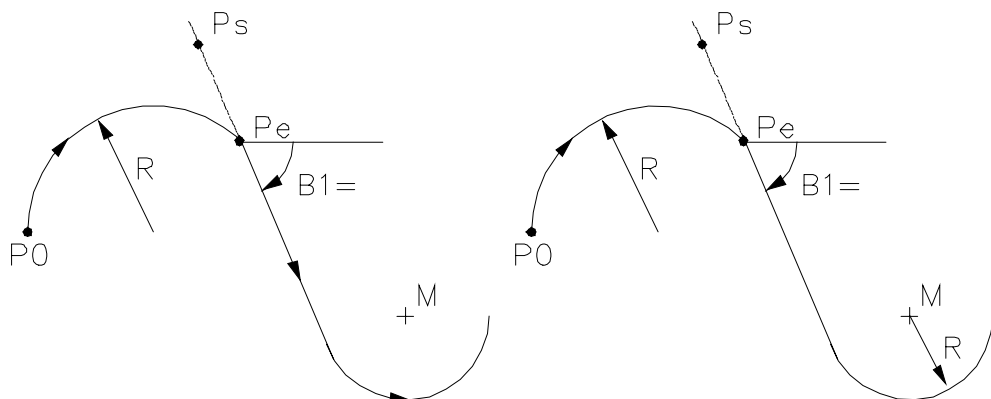
N1	G2/G3	R..	[end point]
N2	G1		[end point]



N1	G2/G3	R..	[end point]
N2	G1	R1=0	

or			
N1	G2/G3	R..	[end point]
N2	G1	B1=..	{R1=0} {J1=1/2}

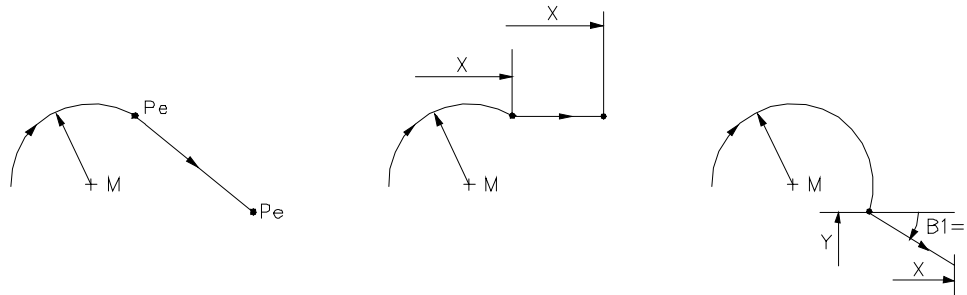
or			
N1	G2/G3	R..	[end point]
N2	G1	X.. Y..	I1=0 {R1=0} {J1=1/2}



N1	G2/G3	R..	[end point]
N2	G1	B1=.. X.. Y..	I1=0 or I1=.. {R1=0} {J1=1/2}

Start point from N1 is not known

If the start point from N1 is not known, the centre point coordinates instead of the radius have to be programmed in block N1. So this block reads:



N1 G2/G3 I.. J.. [end point]

Block N2 from the mentioned cases remains the same.

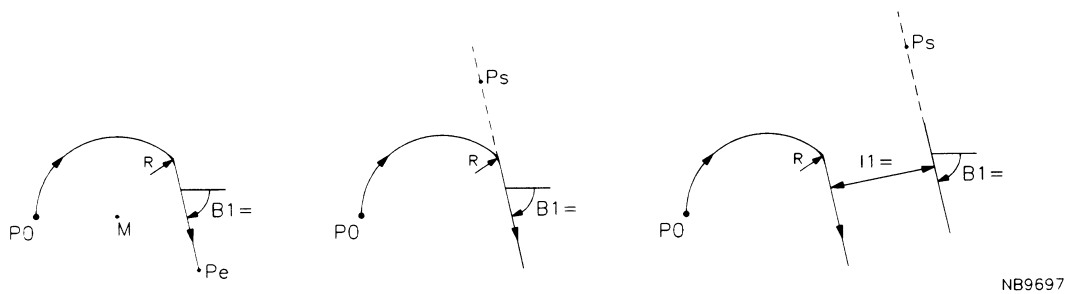
13.1.11 Rounding between intersecting circle and line

To insert a rounding between an intersecting circle and a line.

Notice that the direction of rotation of the rounding is opposite to that of the programmed circle.

calculated intersection point

Start point from N1 is known

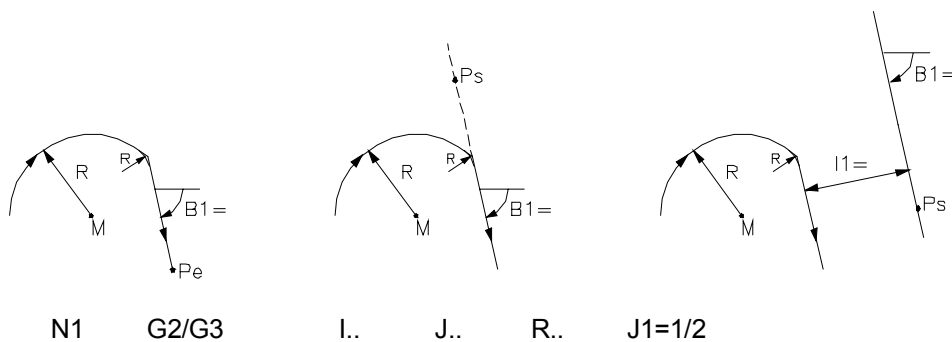


N1	G2/G3	I..	J..	J1=1/2
N2	G3/G2	R..		
N3	G1	etc.		

Refer to the section for calculating the intersection point for the formats of block N3

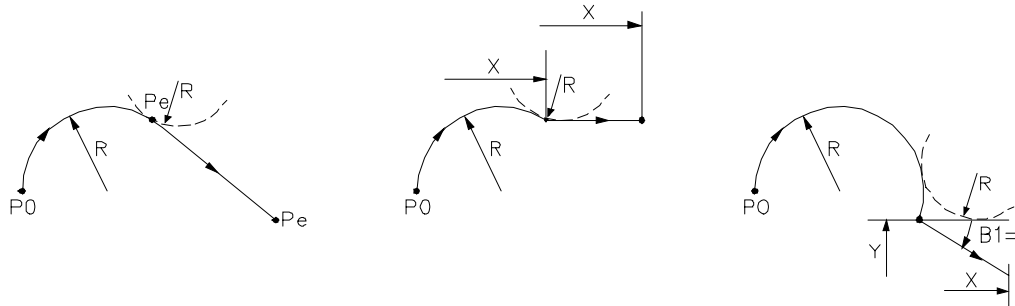
Start point from N1 is not known

If the start point is not known, the value (R-word) should also be programmed in the block N1, thus this block reads:



N1	G2/G3	I..	J..	R..	J1=1/2
----	-------	-----	-----	-----	--------

Refer to the section with the known start point for the formats of block N3

Programmed intersecting point**start point from N1 is known**

N1	G2/G3	R..	[end point]
N2	G3/G2	R..	
N3	G1	etc.	

Refer to the section for programming the intersection point for the formats of block N3.

Note: A rounding can only be inserted if both the circle and the line are programmed with the endpoint, as indicated in the first format.

start point from N1 is not known

If the start point from N1 is not known, the centre point coordinates instead of the radius have to be programmed in block N1. So this block reads:

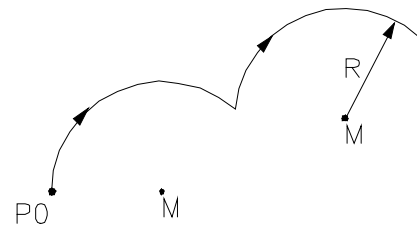
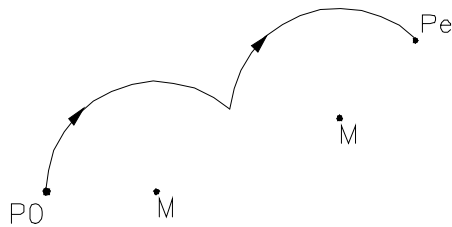
N1	G2/G3	I..	J..	[end point]
N2	G3/G2	R..		

Refer to the section with the known start point for the formats of block N3..

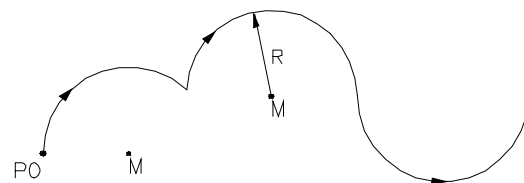
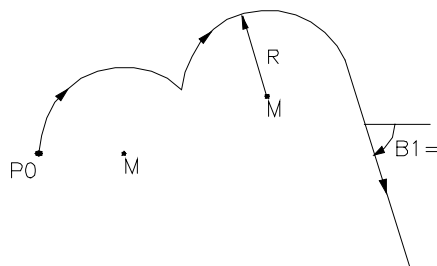
13.1.12 Intersecting point between two circles

To calculate the intersection point between two circles

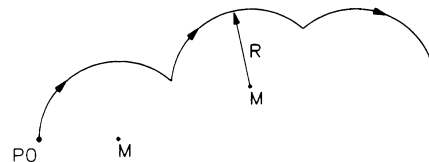
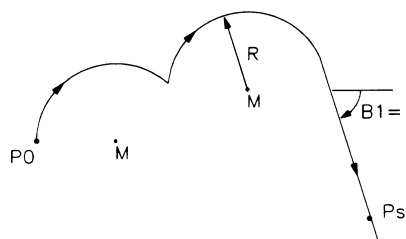
start point from N1 is known



N1	G2/G3	I..	J..	J1=1/2	
N2	G2/G3	I..	J..	X..	Y..
or					
N1	G2/G3	I..	J..	J1=1/2	
N2	G2/G3	I..	J..	R..	



N1	G2/G3	I..	J..	J1=1/2	
N2	G2/G3	I..	J..	R..	R1=0

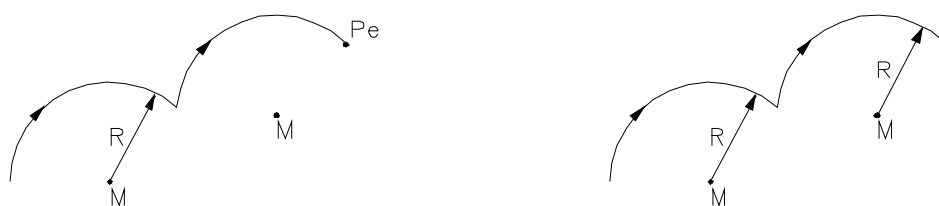


NB9701

N1	G2/G3	I..	J..	J1=1/2	
N2	G2/G3	I..	J..	R..	J1=1/2

start point from N1 is not known

If the start point is not known, the radius value (R-word) should also be programmed in the block N1, thus this block reads:



N1 G2/G3 I.. J.. R.. J1=1/2

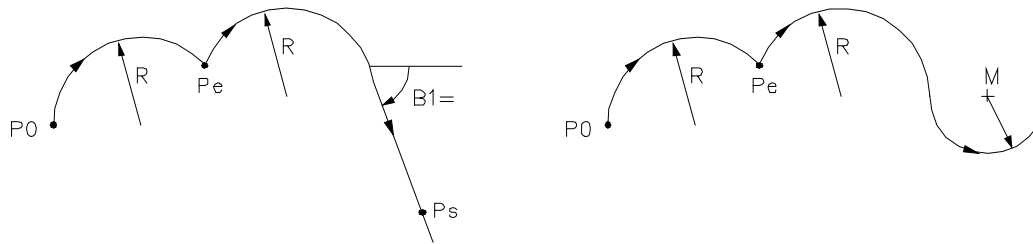
The N2 block remains unchanged.

13.1.13 Intersection point between two circles programmed as end point

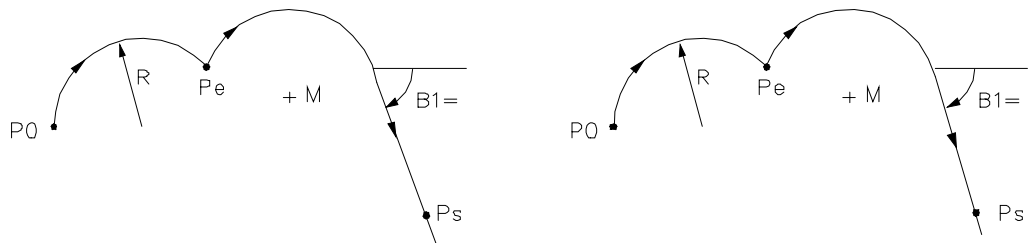
The intersection point between two circles is sometimes shown in the drawing and can be programmed directly. This point is assumed to be the start point of the next movement.

start point from N1 is known

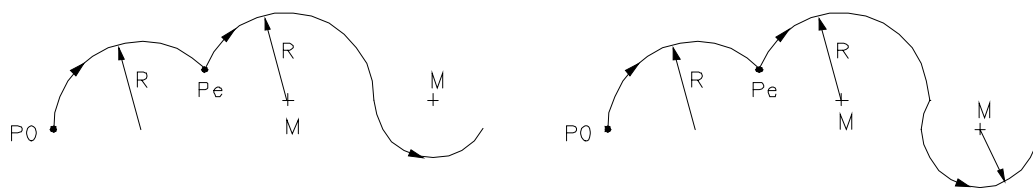
If the start point is not known, the radius value (R-word) should also be programmed in the block N1, thus this block reads:



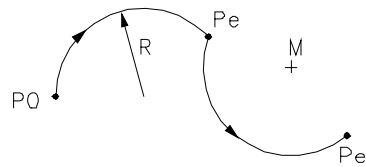
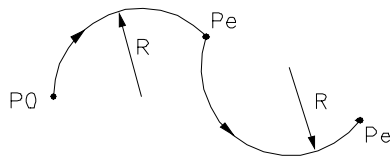
N1	G2/G3	R..	[end point]
N2	G2/G3	R..	R1=0



N1	G2/G3	R..	[end point]
N2	G2/G3	I..	J.. {R1=0} {J1=1/2}



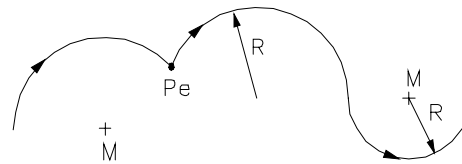
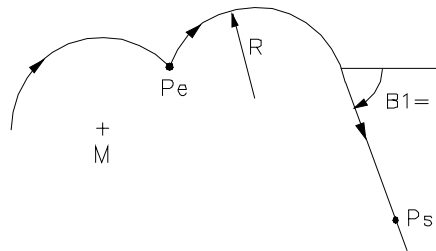
N1	G2/G3	R..	[end point]
N2	G2/G3	I..	J.. R.. {R1=0} {J1=1/2}



N1	G2/G3	R..	[end point]
N2	G2/G3	R..	[end point]
or			
N1	G2/G3	R..	[end point]
N2	G2/G3	I..	J.. [end point]

start point from N1 is not known

If the start point from N1 is not known, the centre point coordinates instead of the radius have to be programmed in block N1. So this block reads:

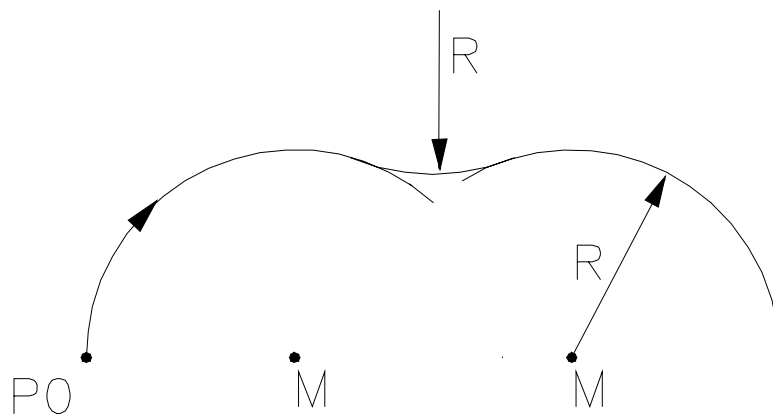


N1	G2/G3	I..	J..	[end point]
----	-------	-----	-----	-------------

Block N2 from the mentioned cases remains the same.

13.1.14 Rounding between two intersecting circles

calculated intersection point
start point from N1 is known



N1	G2/G3	I..	J..	J1=1/2
N2	G3/G2	R..		
N3	G2/G3	etc.		

Refer to the section for calculating the intersection point for the formats of block N3.

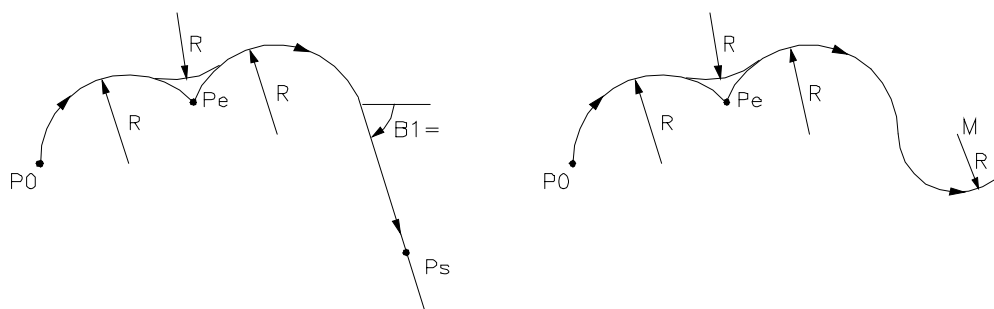
start point from N1 is not known

If the start point is not known, the radius value (R-word) should also be programmed in the block N1, thus this block reads:

N1	G2/G3	I..	J..	R..	J1=1/2
N2	G3/G2	R..			
N3	G2/G3	etc.			

Refer to the section with the known start point for the formats of block N3..

**programmed Intersection point
start point from N1 is known**



N1	G2/G3	R..	[End point]
N2	G3/G2	R..	
N3	G2/G3	etc.	

Refer to the section for programming the intersection point for the formats of block N3.

Start point from N1 is not known

If the start point from N1 is not known, the centre point coordinates instead of the radius have to be programmed in block N1. So this block reads:

N1	G2/G3	I..	J..	[end point]
N2	G3/G2	R..		
N3	G2/G3	etc.		

Refer to the section with the known start point for the formats of block N3.

13.2 Point of tangency

13.2.1 Point of tangency indicator (R1=)

A special word R1=0 is used to indicate that a geometric element is tangent to the next one (connecting circles are disregarded), thus:

line tangent to circle

circle tangent to line

circle tangent to circle.

The word R1=0 is written in the block with the first element.

The point of tangency is chosen in such a way that the tool path is continue, that is to say the tool does not move backwards.

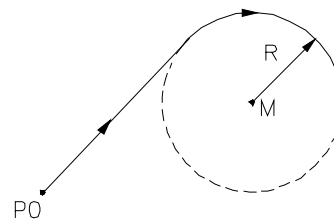
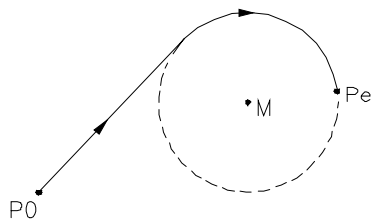
13.2.2 Tangent line and circle

To calculate the point of tangency between line and circle

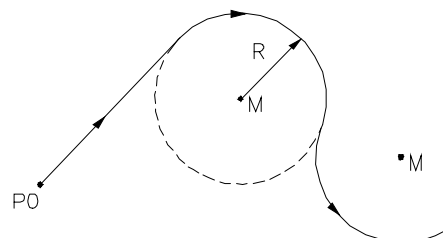
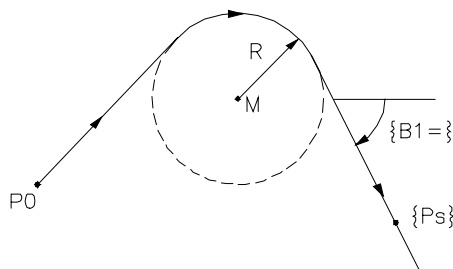
If the start point of the line is known, two cases must be considered:

only the start point from N1 is known

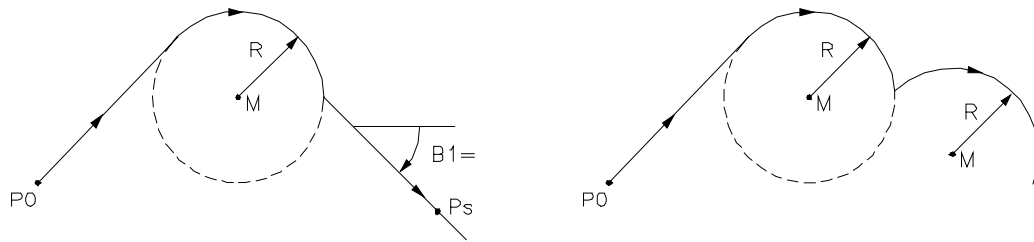
Its centre point coordinates and the radius or end point defines the circle. The following formats for line and circle can be used:



N1	G1	R1=0			
N2	G2/G3	I..	J..	X..	Y..
or					
N1	G1	R1=0			
N2	G2/G3	I..	J..	R..	



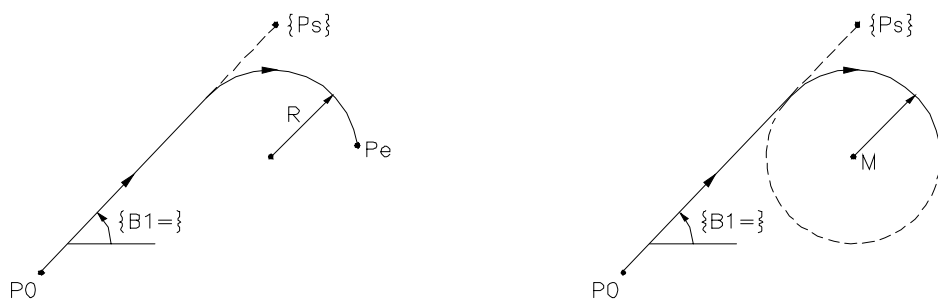
N1	G1	R1=0			
N2	G2/G3	I..	J..	R..	R1=0



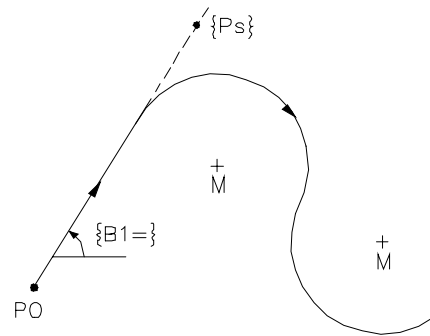
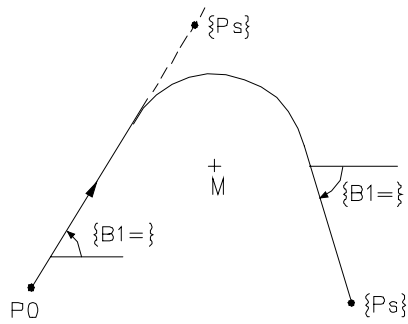
N1	G1	R1=0			
N2	G2/G3	I..	J..	R..	J1=1/2

start point from N1 and the angle with the axis or a support point on the line is known

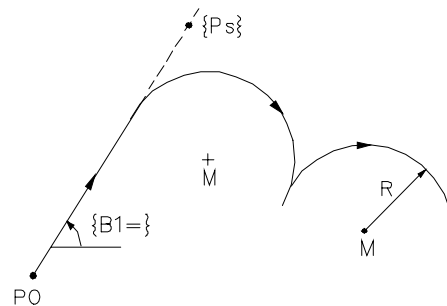
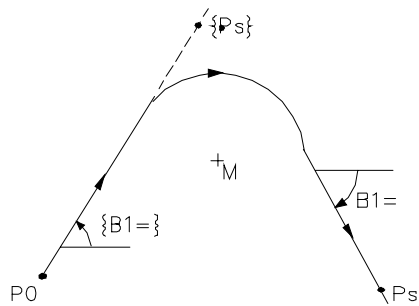
From the circle in block N2 either the radius or the centre point coordinates must be calculated by the control. In this case the following formats are available:



N1	G1	R1=0	B1=...		
N2	G2/G3	R..	X..	Y..	
or					
N1	G1	R1=0	B1=...		
N2	G2/G3	I..	J..		
or					
N1	G1	R1=0	X..	Y..	I1=0
N2	G2/G3	R..	X..	Y..	
or					
N1	G1	R1=0	X..	Y..	I1=0
N2	G2/G3	I..	J..		



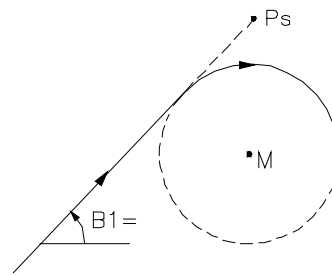
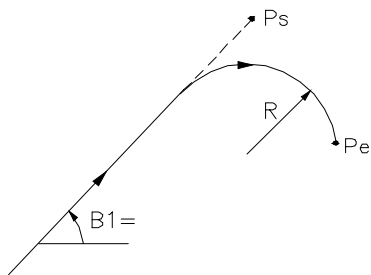
N1	G1	R1=0	B1=...		
N2	G2/G3	I..	J..	R1=0	
or					
N1	G1	R1=0	X..	Y..	I1=0
N2	G2/G3	I..	J..	R1=0	



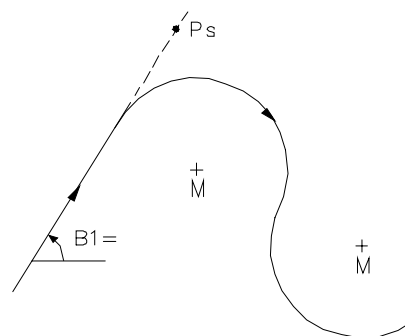
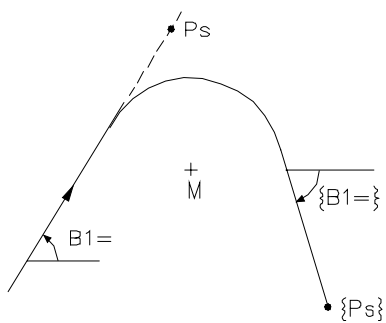
N1	G1	R1=0	B1=...		
N2	G2/G3	I..	J..	J1=1/2	
or					
N1	G1	R1=0	X..	Y..	I1=0
N2	G2/G3	I..	J..	J1=1/2	

start point from N1 is not known

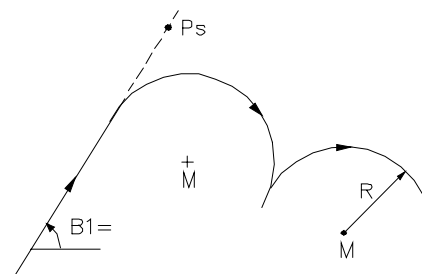
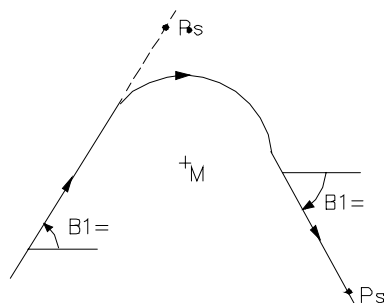
If the start point of the line is not known, the angle with the main axis and a support point on the line have to be programmed. For block N2 the formats from the second case are used, thus:



N1	G1	R1=0	B1=..	X..	Y..	I1=0 or I1=..
N2	G2/G3	R..	X..	Y..		
or						
N1	G1	R1=0	B1=..	X..	Y..	I1=0 or I1=..
N2	G2/G3	I..	J..			



N1	G1	R1=0	B1=..	X..	Y..	I1=0 or I1=..
N2	G2/G3	I..	J..	R1=0		



N1	G1	R1=0	B1=..	X..	Y..	I1=0 or I1=..
N2	G2/G3	I..	J..	J1=1/2		

13.2.3 Continuous connecting circle between tangent line and circle

To insert a connecting circle between a tangent line and a circle. Only one connecting circle is possible. Its direction of rotation is opposite to the direction of rotation on the tangent circle.



start point from N1 is known

N1	G1	R1=0	{B1=..}
N2	G2/G3	R..	
N3	G3/G2	etc.	
or			
N1	G1	R1=0	{X.. Y.. I1=0}
N2	G2/G3	R..	
N3	G3/G2	etc.	

Refer to the previous sections with a unknown start point from N1 for the formats of block N3

start point from N1 is not known

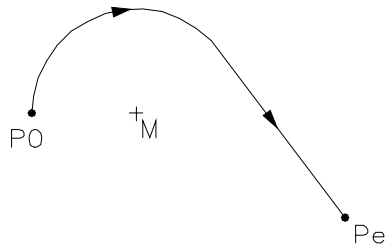
N1	G1		R1=0	B1=..	X..	Y..	I1=0 or I1=..
N2	G2/G3	R..					
N3	G3/G2	etc.					

Refer to the section with unknown start point for the formats of block N3

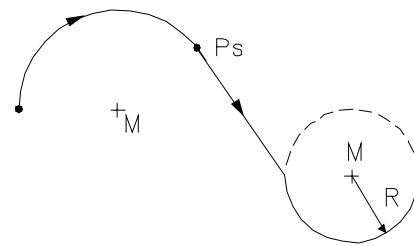
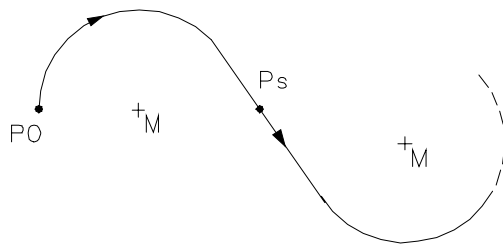
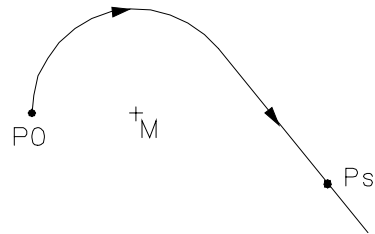
13.2.4 Tangent circle and line

To calculate the point of tangency between circle and line

start point from N1 and the end point or a support point of the line is known

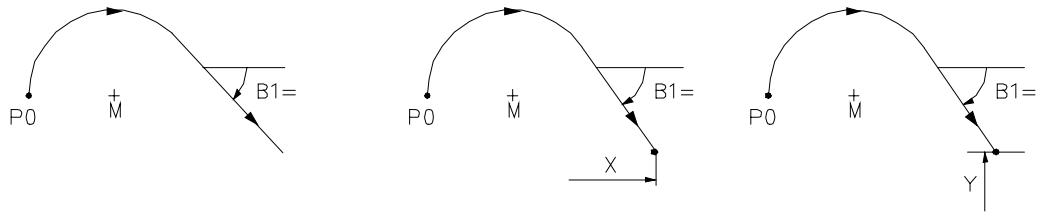


N1	G2/G3	I..	J..	R1=0	
N2	G1		X..	Y..	
or					
N1	G2/G3	I..	J..	R1=0	
N2	G1		X..	Y..	I1=0

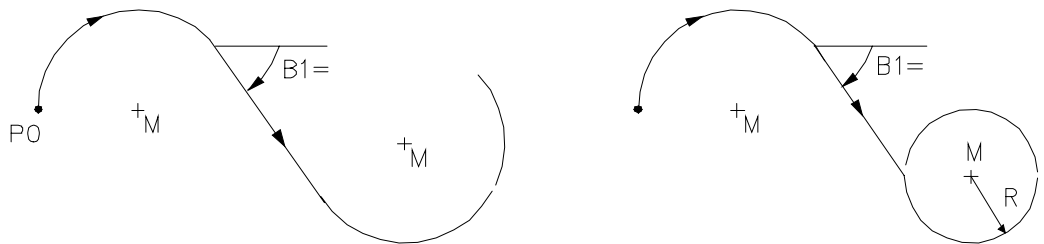


N1	G2/G3	I..	J..	R1=0	
N2	G1	X..	Y..	I1=0	R1=0
or					
N1	G2/G3	I..	J..	R1=0	
N2	G1	X..	Y..	I1=0	J1=1/2

start point from N1 and the angle the line makes with the axis are known

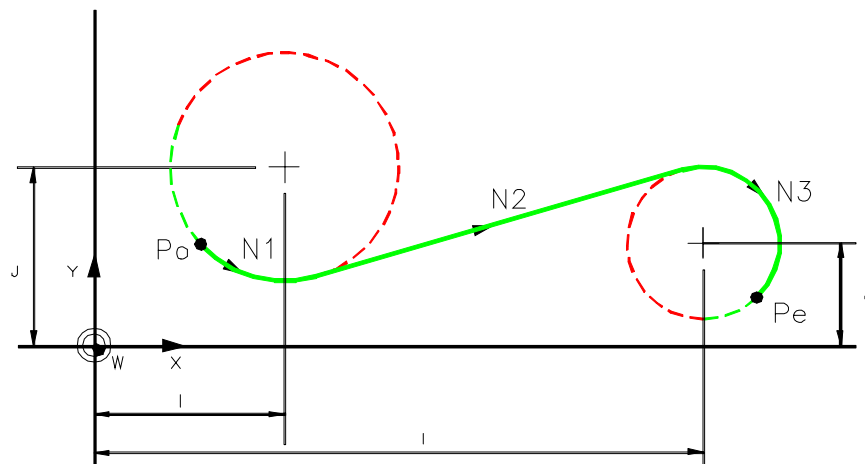


N1	G2/G3	I..	J..	R1=0
N2	G1	B1=..		
or				
N1	G2/G3	I..	J..	R1=0
N2	G1	B1=..	X..	
or				
N1	G2/G3	I..	J..	R1=0
N2	G1	B1=..	Y..	



N1	G2/G3	I..	J..	R1=0
N2	G1	B1=..	R1=0	
or				
N1	G2/G3	I..	J..	R1=0
N2	G1	B1=..	J1=1/2	

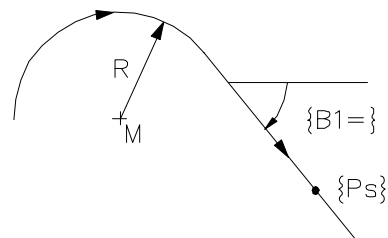
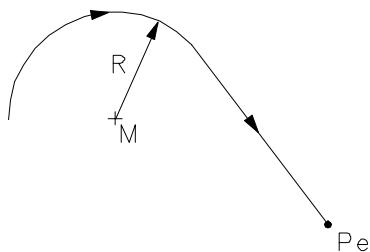
common tangent line of two circles



N1	G2/G3	I..	J..	R1=0
N2	G1	R1=0		
N3	G2/G3	I..	J..	R..
or				
N1	G2/G3	I..	J..	R1=0
N2	G1	R1=0		
N3	G2/G3	I..	J..	X.. Y..

Start point from N1 is not known

If the start point is not known, the radius value (R-word) should also be programmed in the block N1, thus this block reads:

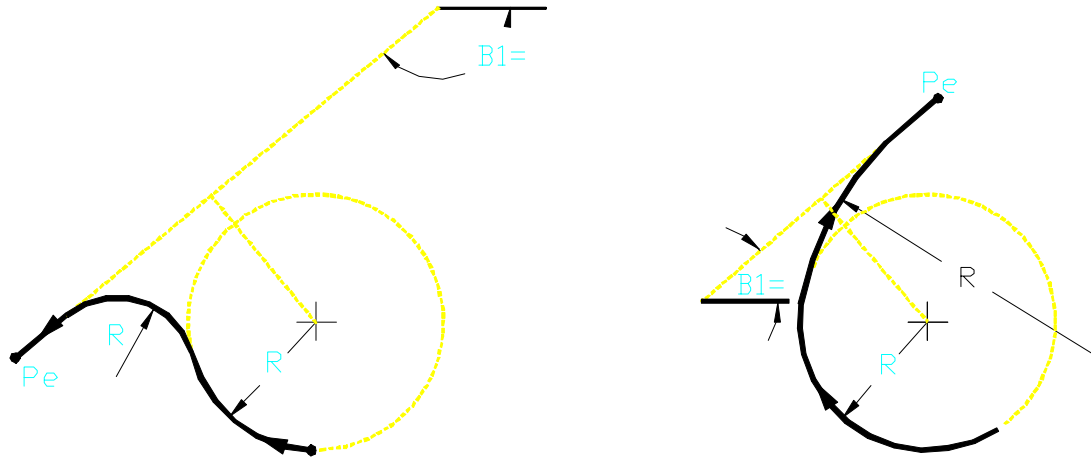


N1	G2/G3	I..	J..	R..	R1=0
----	-------	-----	-----	-----	------

The other blocks from the mentioned cases remain the same.

13.2.5 Continuous connection circle between tangent circle and line

To insert a connecting circle between a tangent circle and line. Only one connecting circle is possible. Its direction of rotation is opposite to the direction of rotation on the tangent circle



start point from N1 is known

N1	G2/G3	I..	J..	R1=0
N2	G3/G2	R..		
N3	G1	etc.		

Refer to the previous sections with a known start point for the formats of block N3.

Start point from N1 is not known

If the start point is not known, the radius value (R-word) should also be programmed in the block N1, thus this block reads:

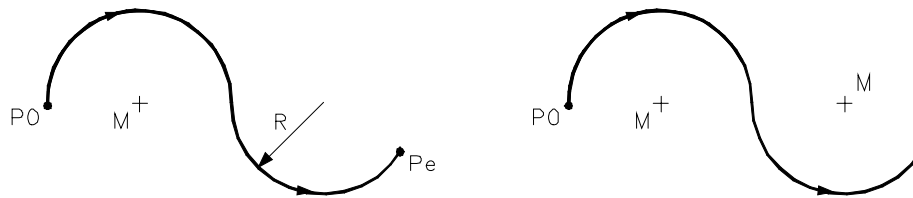
N1	G2/G3	I..	J..	R..	R1=0
N2	G3/G2	R..			
N3	G1	etc.			

Refer to the section with unknown start point for the formats of block N3.

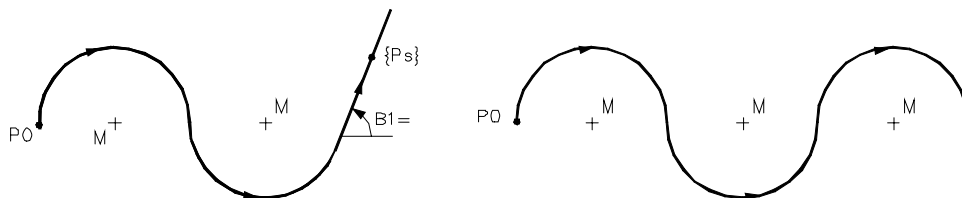
13.2.6 Tangent circle and line

To calculate the point of tangency between circle and line

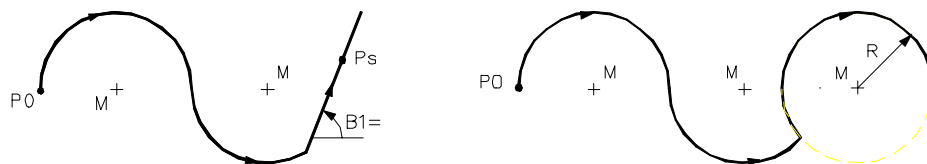
start point from N1 is known



N1	G2/G3	I..	J..	R1=0
N2	G2/G3	R..	X..	Y..
or				
N1	G2/G3	I..	J..	R1=0
N2	G2/G3	I..	J..	



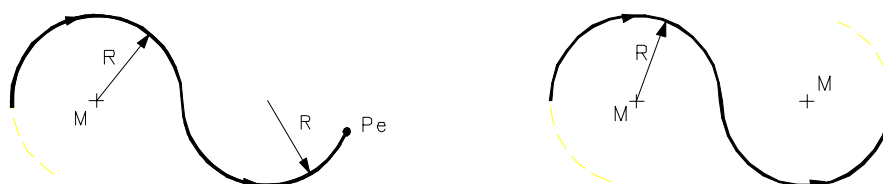
N1	G2/G3	I..	J..	R1=0
N2	G2/G3	I..	J..	R1=0



N1	G2/G3	I..	J..	R1=0
N2	G2/G3	I..	J..	J1=1/2

Start point from N1 is not known

If the start point is not known, the radius value (R-word) should also be programmed in the block N1, thus this block reads:



N1	G2/G3	I..	J..	R..	R1=0
----	-------	-----	-----	-----	------

The other blocks from the mentioned cases remain the same.

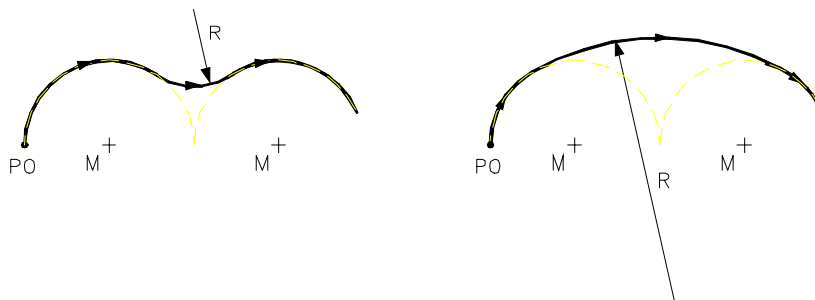
13.2.7 Continuous connecting circle between two tangent circles

To insert a connecting circle between two tangent circles. The connecting circle can be outside both circles or surround them. The programming of the direction of rotation on the three circles is:

1. Connecting circle tangent on the outside of both circles
G2, G3, G2 or G3, G2, G3
2. Connecting circle surrounds both circles
G2, G2, G2 or G3, G3, G3

For both cases the following formats are available. The right combination of the direction of rotation on the three circles has to be chosen by the programmer..

start point from N1 is known



N1	G2/G3	I..	J..	R1=0
N2	G3/G2	R..		
N3	G2/G3	etc.		

Refer to the previous sections with a known start point for the formats of block N3.

Start point from N1 is not known

N1	G2/G3	I..	J..	R..	R1=0
----	-------	-----	-----	-----	------

The other blocks from the mentioned cases remain the same.

13.3 Continuous connecting circle between elements which do not meet

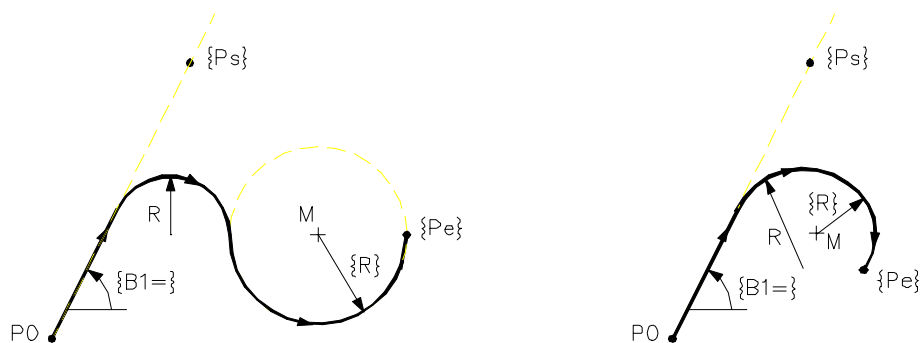
13.3.1 Line and circle

To insert a connecting circle between a line which does not meet a circle. Two connecting circles are possible:

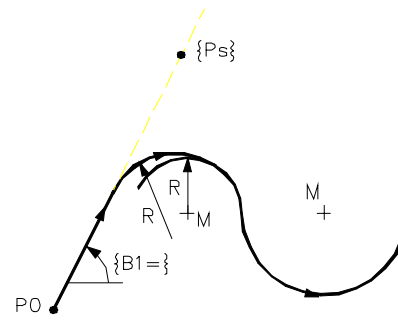
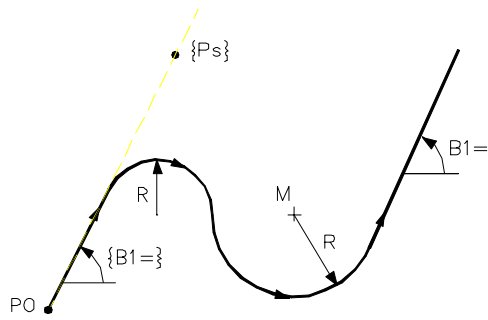
1. Connecting circle is tangent on the outside of the circle
G1, G2, G3 or G1, G3, G2
2. Connecting circle surrounds the circle
G1, G2, G2 or G1, G3, G3

For both cases the following formats are available. The right combination of the direction of rotation on the circles has to be chosen by the programmer.

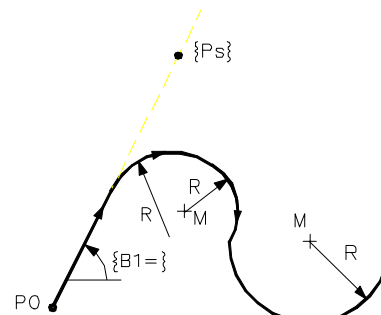
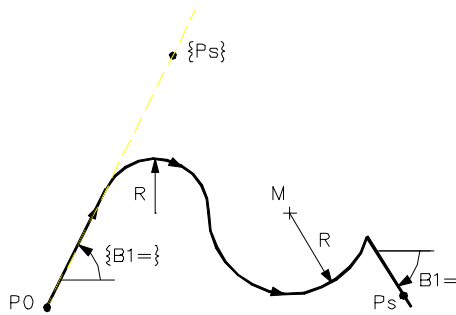
start point from N1 is known



N1	G1	B1=..			
N2	G3/G2	R..			
N3	G2/G3	I..	J..	X..	Y..
or					
N1	G1	B1=..			
N2	G3/G2	R..			
N3	G2/G3	I..	J..	R..	
or					
N1	G1	X..	Y..	I1=0	
N2	G3/G2	R..			
N3	G2/G3	I..	J..	X..	Y..
or					
N1	G1	X..	Y..	I1=0	
N2	G3/G2	R..			
N3	G2/G3	I..	J..	R..	



N1	G1	B1=..			
N2	G3/G2	R..			
N3	G2/G3	I..	J..	R..	R1=0
or					
N1	G1	X..	Y..	I1=0	
N2	G3/G2	R..			
N3	G2/G3	I..	J..	R..	R1=0



N1	G1	B1=..			
N2	G3/G2	R..			
N3	G2/G3	I..	J..	R..	J1=1/2
or					
N1	G1	X..	Y..	I1=0	
N2	G3/G2	R..			
N3	G2/G3	I..	J..	R..	J1=1/2

start point from N1 is not known

N1	G1	B1=...	X..	Y..	I1=0 or I1=..
----	----	--------	-----	-----	---------------

The other blocks from the mentioned cases remain the same.

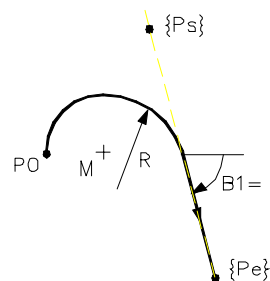
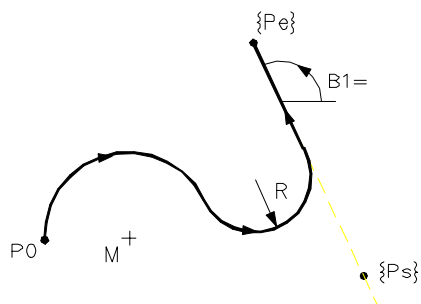
13.3.2 Circle and line

To insert a connecting circle between a circle and a line which do not meet each other. Two connecting circles are possible:

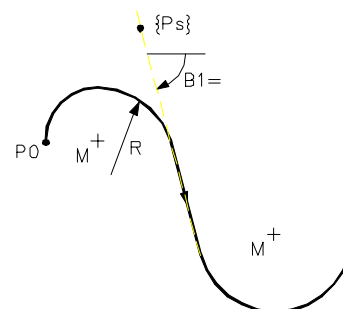
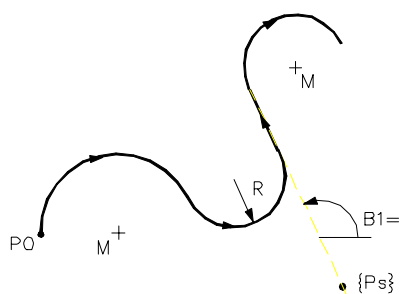
1. Connecting circle is tangent on the outside of the circle
G2, G3, G1 or G3, G2, G1
2. Connecting circle surrounds the circle
G2, G2, G1 or G3, G3, G1

For both cases the following formats are available. The right combination of the direction of rotation on the circles has to be chosen by the programmer.

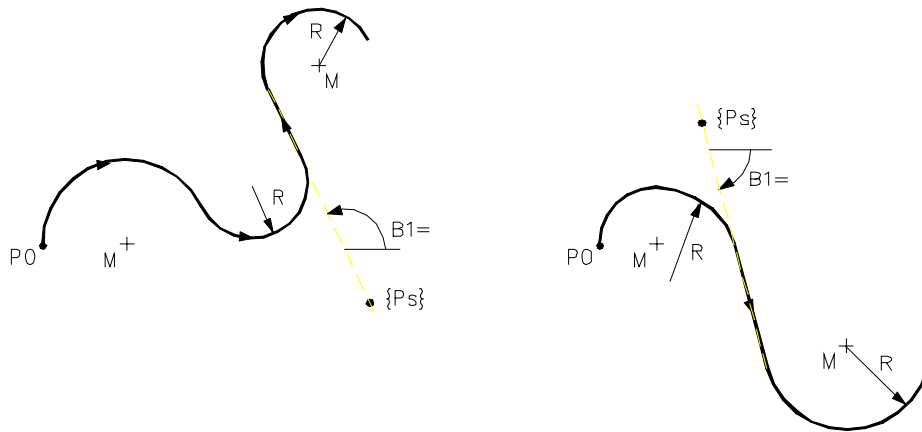
Start point from N1 is known



N1	G2/G3	I..	J..		
N2	G3/G2	R..			
N3	G1	B1=..	X..	Y..	
or					
N1	G2/G3	I..	J..		
N2	G3/G2	R..			
N3	G1	B1=..	X..	Y..	I1=0



N1	G2/G3	I..	J..		
N2	G3/G2	R..			
N3	G1	B1=..	X..	Y..	I1=0 R1=0



N1	G2/G3	I..	J..			
N2	G3/G2	R..				
N3	G1	B1=..	X..	Y..	I1=0	J1=1/2

start point from N1 is not known

N1	G2/G3	I..	J..	R..
----	-------	-----	-----	-----

The other blocks from the mentioned cases remain the same.

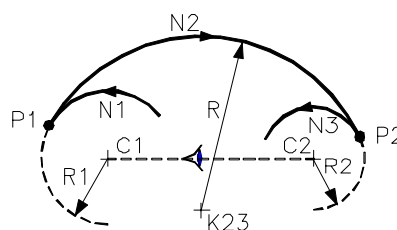
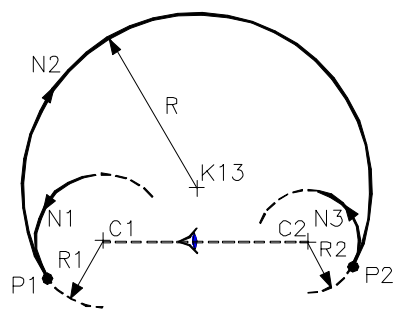
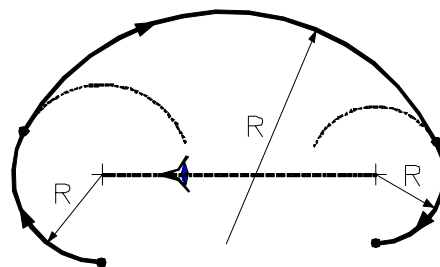
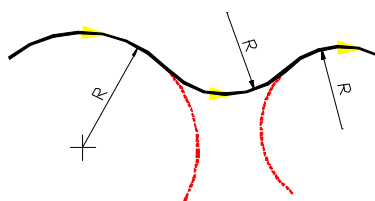
13.3.3 Two circles outside each other

To insert a connecting circle between two circles outside each other which do not meet. The direction of rotation on the three circles indicates the type of connecting circle, thus:

1. Connecting circle tangent on the outside of both circles
G2, G3, G2 or G3, G2, G3
2. Connecting circle surrounds both circles
G2, G2, G2 or G3, G3, G3
3. Connecting circle is tangent on the outside of the first circle and surrounds the second circle
G2, G3, G3 or G3, G2, G2
4. Connecting circle surrounds the first circle and is tangent on the outside of the second circle
G2, G2, G3 or G3, G3, G2

For all four cases the following formats are available. Any combination of the direction of rotation on the three circles is possible:

Start point from N1 is known



N1	G2/G3	I..	J..		
N2	G3/G2	R..			
N3	G2/G3	I..	J..	X..	Y..
or					
N1	G2/G3	I..	J..		
N2	G3/G2	R..			
N3	G2/G3	I..	J..	R..	{R1=0} {J1=1/2}

start point from N1 is not known

N1 G2/G3 I.. J.. R..

The other blocks from the mentioned cases remain the same.

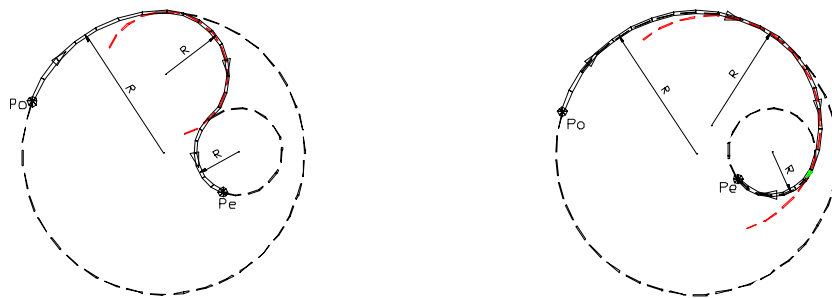
13.3.4 One circle inside the other one

To insert a connecting circle between a circle inside the other one which do not meet. The direction of rotation on the three circles indicates the type of connecting circle, thus:

1. Connecting circle tangent on the outside of inner circle
G2, G2, G3 or G3, G3, G2
2. Connecting circle tangent on the outside of inner circle
G2, G2, G2 or G3, G3, G3

For both cases the following formats are available:

start point from N1 is known



N1	G2/G3	I..	J..		
N2	G2/G3	R..			
N3	G2/G3	I..	J..	X..	Y..
or					
N1	G2/G3	I..	J..		
N2	G2/G3	R..			
N3	G2/G3	I..	J..	R..	{R1=0} {J1=1/2}

Start point from N1 is not known

N1 G2/G3 I.. J.. R..

The other blocks from the mentioned cases remain the same.

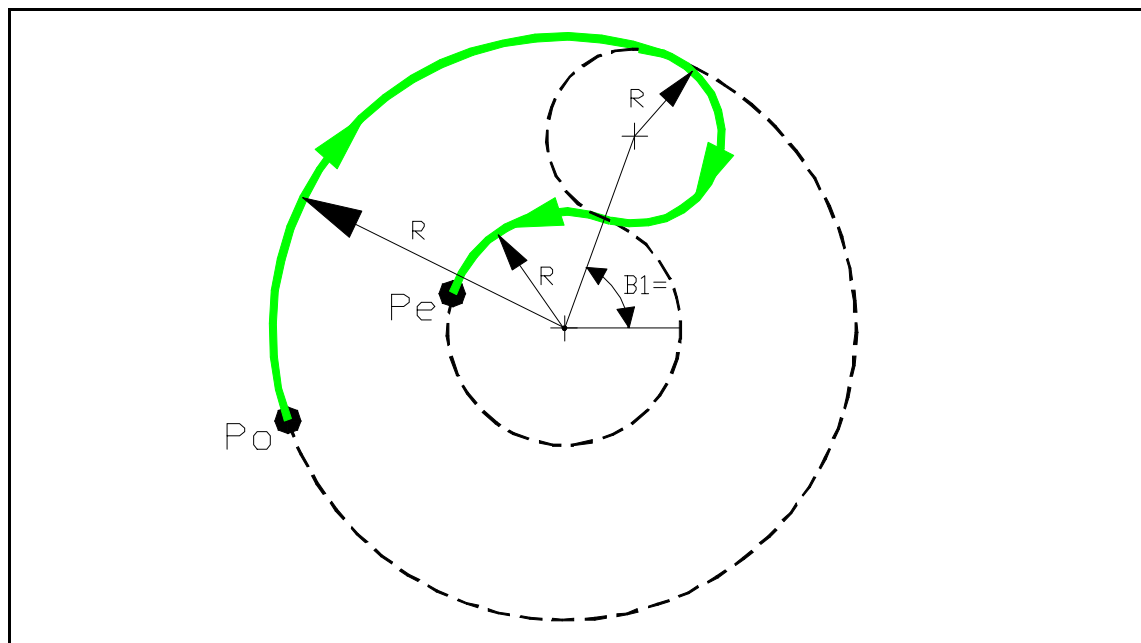
13.3.5 Concentric circles

Two concentric circles are a very special case of one circle inside the other one. In this case the centre points of both circles coincide, so some additional information has to be programmed to let the control calculate the connecting circle.

The word B1=.. which indicate the angle with the main axis of the line through the centre point of the concentric circles and the connecting circle, is used as additional information and has to be inserted in the block with the connecting circle.

For both cases the following formats are available:

start point from N1 and radius of the connecting circle are known



N1	G2/G3	I..	J..	
N2	G2/G3	R..	B1=..	
N3	G2/G3	I..	J..	{R1=0} {J1=1/2}

start point from N1 and radius of the second circle are known

In this case the radius of the connecting circle is calculated by the control.

N1	G2/G3	I..	J..		
N2	G2/G3	B1=..			
N3	G2/G3	I..	J..	X..	Y..

or

N1	G2/G3	I..	J..		
N2	G2/G3	B1=..			
N3	G2/G3	I..	J..	R..	{R1=0} {J1=1/2}

start point from N1 is not known

N1	G2/G3	I..	J..	R..
----	-------	-----	-----	-----

The other blocks from the mentioned cases remain the same.

13.4 G64 Geometric calculations with non-continuous movements

Contents of this format section

- 4.1 Rounding or connecting circle indicator (K1=)
- 4.2 Rounding with intersection points
- 4.3 Rounding between intersecting straight lines
- 4.4 Rounding between intersecting line and circle
- 4.5 Rounding between intersecting circle and line
- 4.6 Rounding between two intersecting circles
- 4.7 Tangent lines (r1=)
- 4.8 Connecting circle between tangent line and circle or v.v.
- 4.9 Connecting circle between a line which do not meet a circle
- 4.10 Connecting circle between circles outside each other
- 4.11 Connecting circle between two circles one inside the other
- 4.12 Connecting circle with two concentric circles

13.4.1 Rounding or connecting circle indicator (K1=)

A special indicator (K1 =) is introduced to program which rounding or connecting circle is to be used.

For a rounding the value of K1= can be 1, 2, 3, or 4. Refer to the section ROUNDING WITH INTERSECTION POINTS for the meaning of these values. If with a rounding a wrong value is programmed, an error message is displayed.

For a connecting circle the indicator K1 = has two digits:

the first digit can have the value 1 or 2

- =1 the left connecting circle
- =2 the right connecting circle

the second digit indicates which connecting circle is meant and can have the values

- =0 with a line tangent to a circle or v.v.
- =0 or 1 with a line which does not meet a circle or v.v.
- =2 to 7 with circular elements

Refer to the proper sections for programming a connecting circle to see the meaning of left and right and of the second digit.

13.4.2 Rounding with intersection points

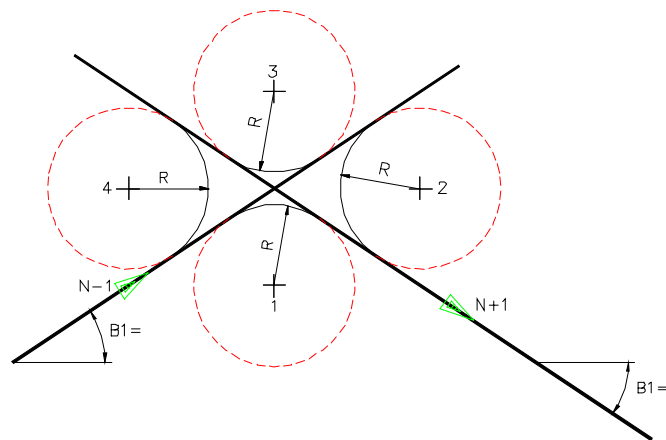
Between two intersecting elements a rounding can be inserted. In general four circles are possible which are numbered 1 to 4 and programmed with the word K1 = . The centre points of the circles with the numbers 1 and 2 are lying at the right from the first geometry element, when looking in the direction of the tool movement.

With the words K1 =2 or K1 =3 the contour intersects itself.

Note: If the word K1= is not programmed, a default value is used. This is K1=1 or K1=4 depending on the direction of movement on the second element.

13.4.3 Rounding between intersecting straight lines

To insert a rounding between two straight lines. The rounding can have any direction of rotation, thus G2 or G3.



start point from N1 is known

N1	G1	B1=..
N2	G2/G3 R..	K1=1/2/3/4
N3	G1	etc.

Refer to the corresponding section with continuous movements for the formats of block N1 and N3.

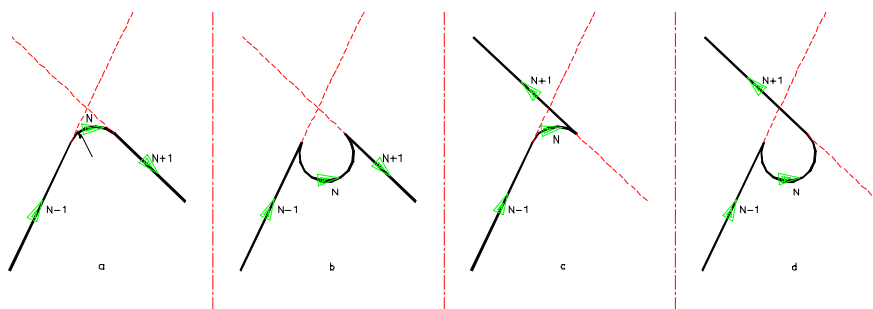
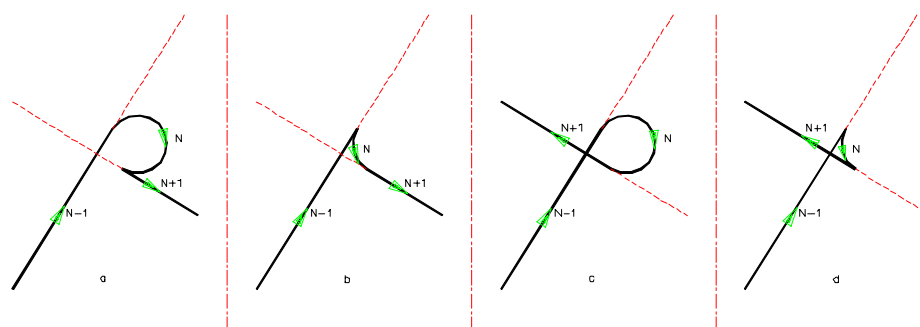
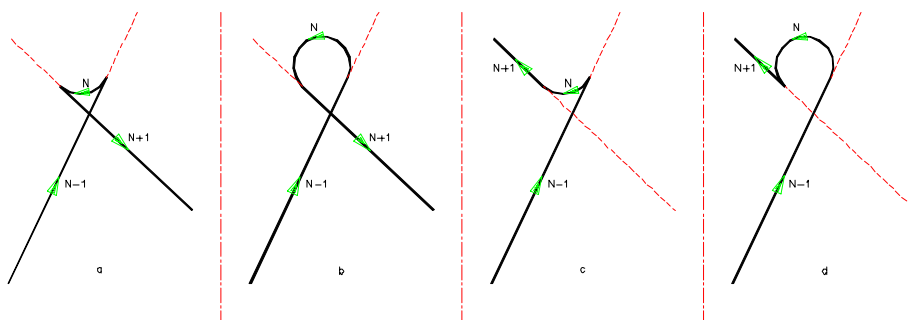
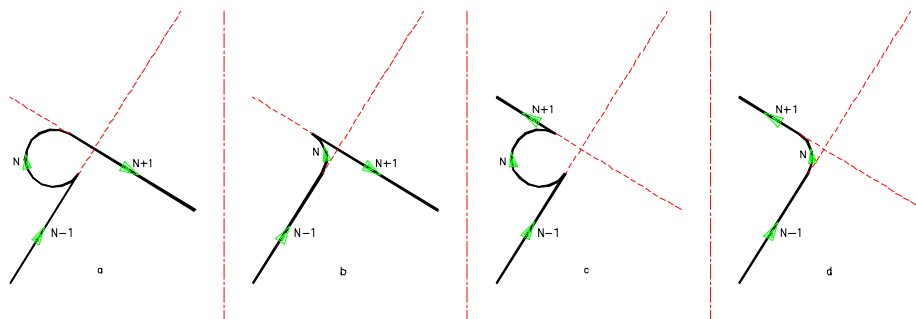
start point from N1 is not known

If the start point from N1 is not known, both the angle and a support point have to be programmed in block N1. So this block reads:

N1	G1	B1=..	X..	Y..	I1=0 or I1=..
N2	G2/G3 R..	K1=1/2/3/4			
N3	G1	etc.			

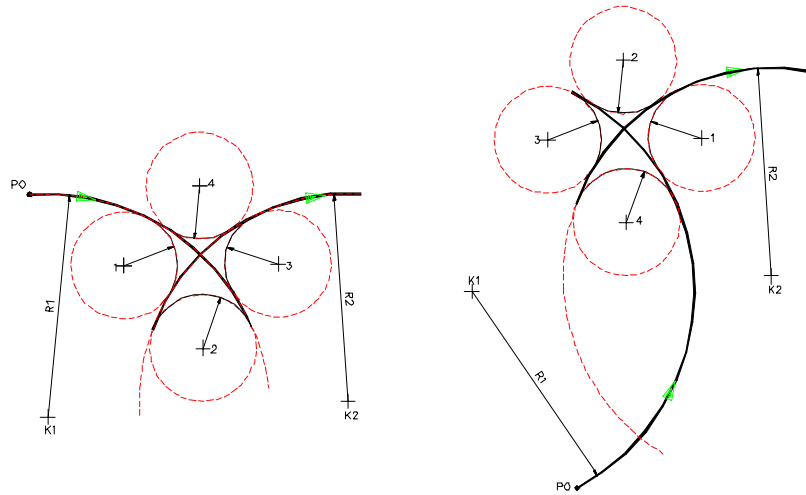
Refer to the corresponding section with continuous movements for the formats of block N1 and N3.

In the following illustrations the possibilities are shown with a circular connection between two straight lines.

A circular connection with $K1=1$ A circular connection with $K1=2$ A circular connection with $K1=3$ A circular connection with $K1=4$

13.4.4 Rounding between intersecting line and circle

To insert a rounding between an intersecting line and a circle. The rounding can have any direction of rotation, thus G2 or G3.



start point from N1 is known

N1	G1	B1=.	J1=1/2
N2	G3/G2	R..	K1=1/2/3/4
N3	G2/G3	etc.	

start point from N1 is not known

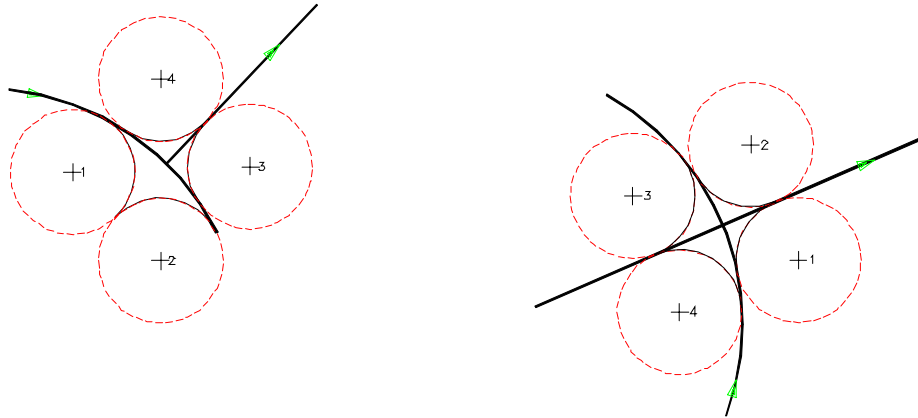
If the start point from N1 is not known, both the angle and a support point have to be programmed in block N1. So this block reads:

N1	G1	B1=.	X..	Y..	I1=0 or I1=.	J1=1/2
N2	G3/G2	R..	K1=1/2/3/4			
N3	G2/G3	etc.				

Refer to the corresponding section with continuous movements for the formats of block N1 and N3.

13.4.5 Rounding between intersecting circle and line

To insert a rounding between an intersecting circle and a line. The rounding can have any direction of rotation, thus G2 or G3.



Start point from N1 is known

N1	G2/G3	I..	J..	J1=1/2
N2	G3/G2	R..	K1=1/2/3/4	
N3	G1	etc.		

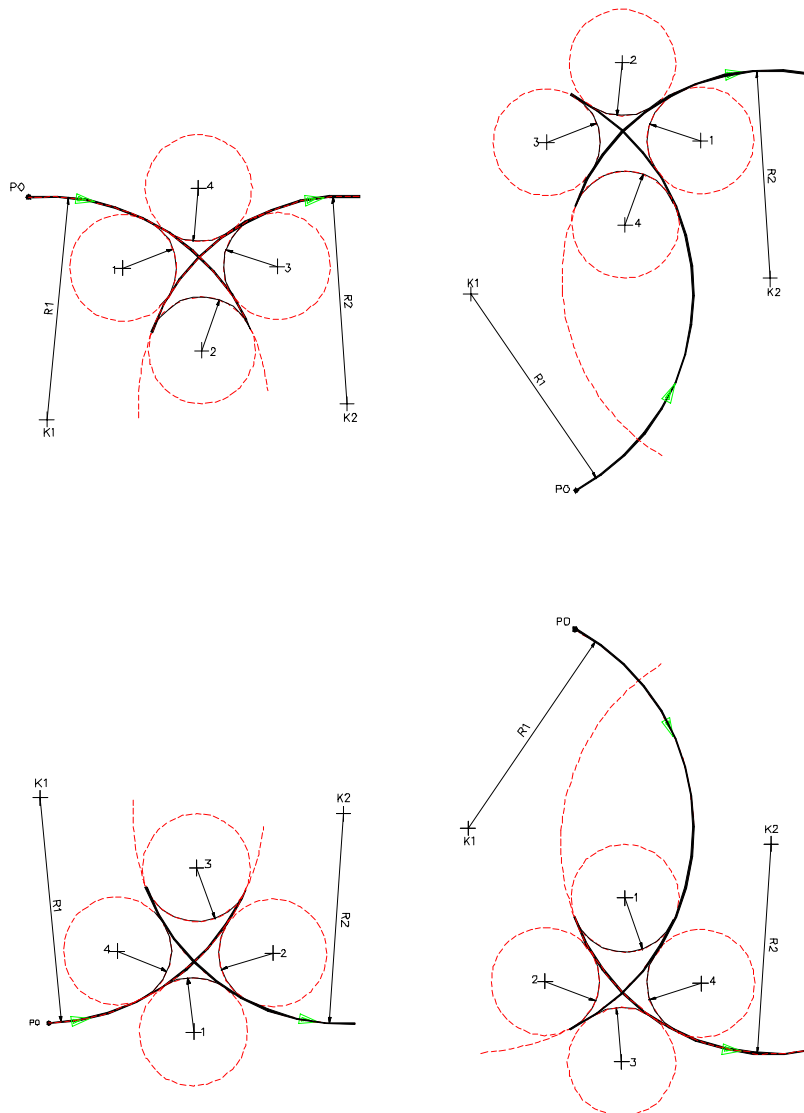
Start point from N1 is not known

N1	G2/G3	I..	J..	R..	J1=1/2
N2	G3/G2	R..	K1=1/2/3/4		
N3	G1	etc.			

Refer to the corresponding section with continuous movements for the formats of block N1 and N3.

13.4.6 Rounding between two intersecting circles

To insert a rounding between two intersecting circles. The rounding can have any direction of rotation, thus G2 or G3.



start point from N1 is known

N1	G2/G3	I..	J..	J1=1/2
N2	G3/G2	R..	K1=1/2/3/4	
N3	G2/G3	etc.		

start point from N1 is not known

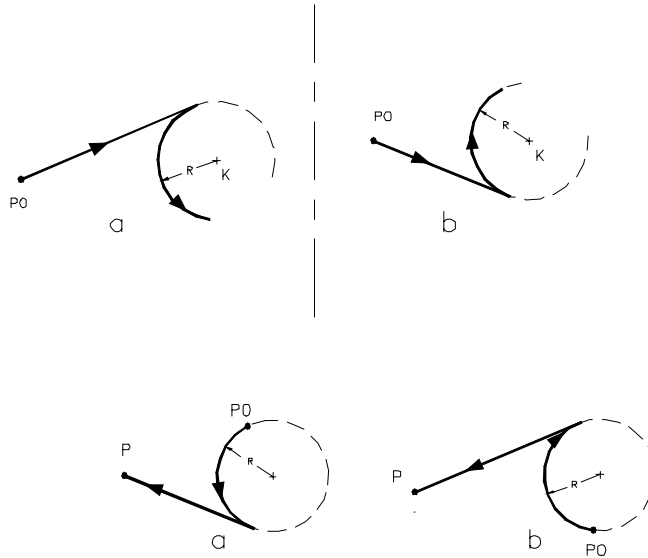
N1	G2/G3	I..	J..	R..	J1=1/2
N2	G3/G2	R..	K1=1/2/3/4		
N3	G2/G3	etc.			

Refer to the corresponding section with continuous movements for the formats of block N1 and N3.

13.4.7 Tangent lines (R1=)

From a point two lines can be drawn tangent to a circle. With the word R1= in the block with the tangent element is indicated which tangent line should be used:

- a. R1=1: the left tangent line
- b. R1=2: the right tangent line



Left and right are determined with a movement from:

- line to circle** by looking from start point to centre point
- circle to line** by looking from centre point to end point

The word R1=1 or R1=2 is programmed in the same way as explained for R1=0 in the continuous section.

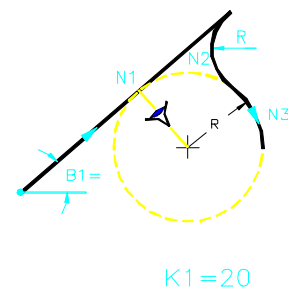
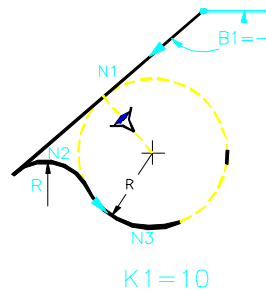
Note: With R1=0 the control determines automatically which tangent line keeps the movement continuous, so the tool does not move backwards.

13.4.8 Connecting circle between a line tangent to a circle or v.v.

If a line is tangent to a circle two connecting circles are possible, one left of the line through the centre of the circle and perpendicular to the line and one to the right of that line.

The left circle is programmed with the indicator K1=10 and the right circle with K1=20.

The formats are:

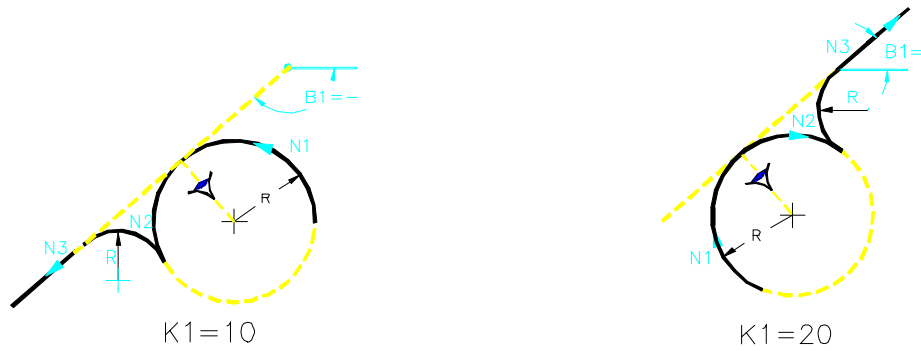
Gerade tangiert an Kreis**start point from N1 is known**

N1	G1		R1=1/2
N2	G2/G3	R..	K1=10/20
N3	G2/G3	etc.	

start point from N1 is not known

N1	G1	B1=..	R1=1/2
N2	G2/G3	R..	K1=10/20
N3	G2/G3	etc.	

Refer to the corresponding section with continuous movements for the formats of block N1 and N3.

Circle tangent to line**start point from N1 is known**

N1	G2/G3	I..	J..	R1=1/2
N2	G2/G3	R..	K1=10/20	
N3	G1	etc.		

start point from N1 is not known

N1	G2/G3	I..	J..	R..	R1=1/2
N2	G2/G3	R..	K1=10/20		
N3	G1	etc.			

Refer to the corresponding section with continuous movements for the formats of block N1 and N3.

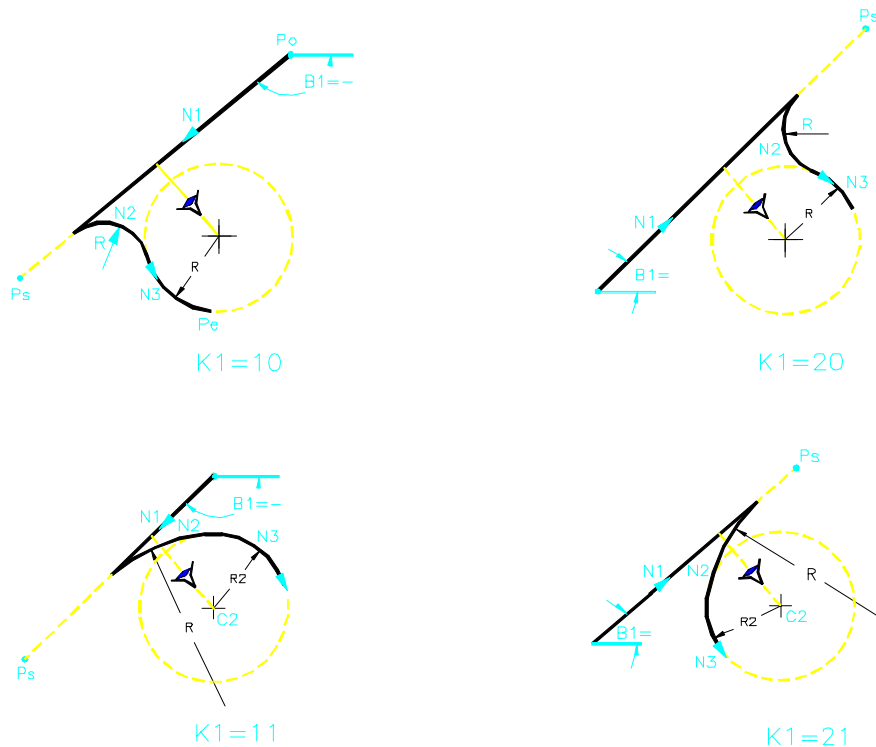
13.4.9 Connecting circle between a line which does not meet a circle

If a line does not meet a circle, two connecting circles are possible on the left of the line through the centre and perpendicular to the line. The same two circles are also possible to the right of that line. One circle touches the circle on the outside. The left circle is programmed with the word K1=10 and the same circle on the right with K1=20.

The second connecting circle surrounds the circle. In this case the left circle is programmed with the word K1=11 and the same circle on the right with K1=21.

The formats are:

Line and circle



start point from N1 is known

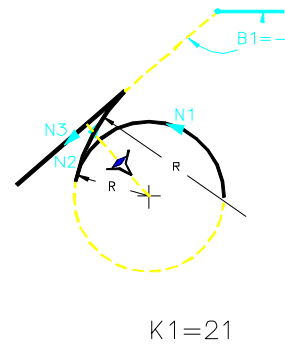
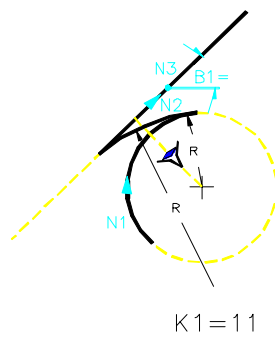
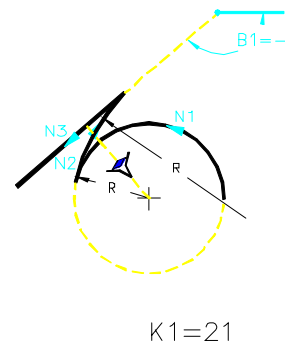
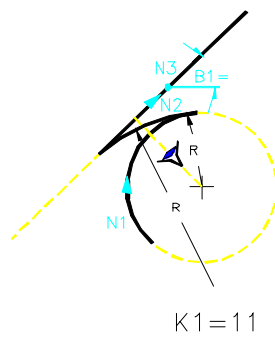
N1	G1	B1=..	{X.. Y.. I1=0}
N2	G2/G3	R..	K1=10/11 or K1=20/21
N3	G2/G3	etc.	

start point from N1 is not known

N1	G1	B1=..	X.. Y.. I1=0
N2	G2/G3	R..	K1=10/11 or K1=20/21
N3	G2/G3	etc.	

Refer to the corresponding section with continuous movements for the formats of block N1 and N3.

Circle and line



start point from N1 is known

N1	G2/G3	I..	J..	R1=1/2
N2	G2/G3	R..	K1=10/11 or K1=20/21	
N3	G1	etc.		

start point from N1 is not known

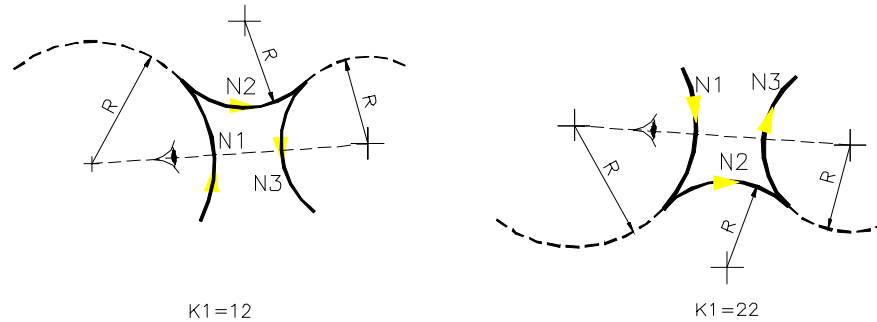
N1	G2/G3	I..	J..	R..	R1=1/2
N2	G2/G3	R..	K1=10/11 or K1=20/21		
N3	G1	etc.			

Refer to the corresponding section with continuous movements for the formats of block N1 and N3.

13.4.10 Connecting circle between circles outside each other

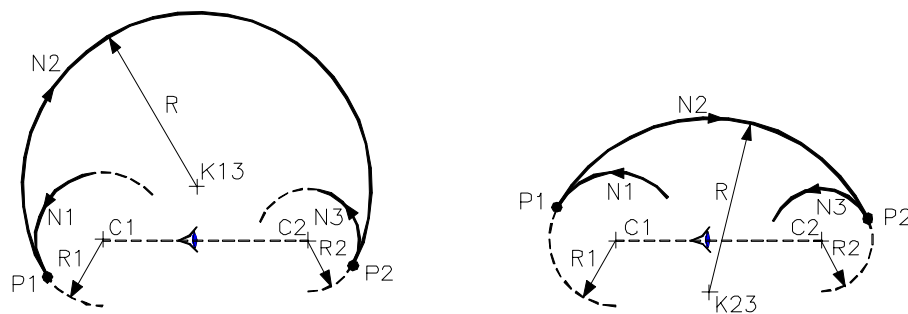
Four types of connecting circles on the left of the line through the centre points are possible with two circles, which do not meet and are outside each other. The first two types are also possible with tangent circles. The same four types can be found on the right of the line through the centres.

The word K1=.. for a connecting circle outside both circles is:



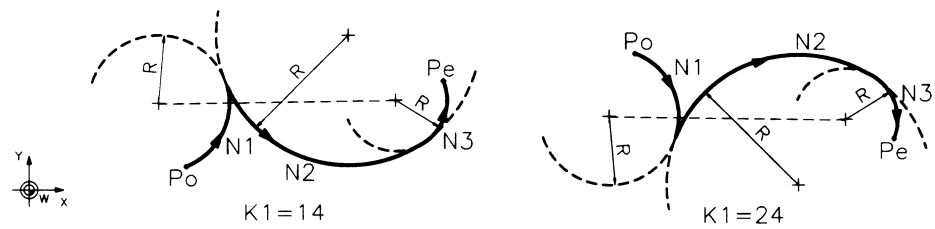
- for the **left** connecting circle: K1 =12
- for the **right** connecting circle: K1 =22

The word K1=.. for a connecting circle **surrounding both circles** is:



- for the **left** connecting circle: K1=13
- for the **right** connecting circle: K1=23

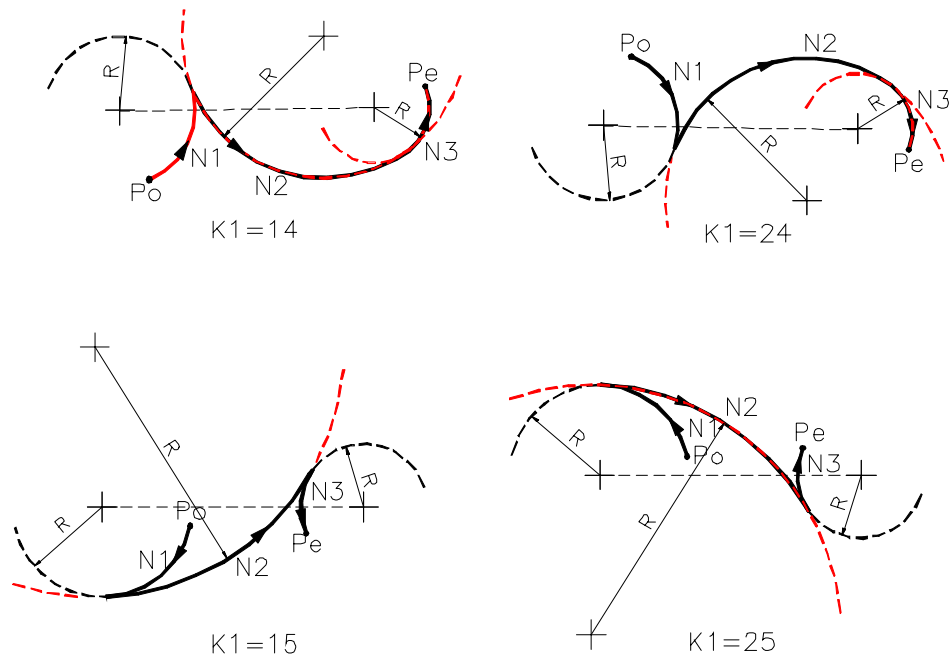
The word K1 =.. for a connecting circle **outside the first circle** is:



(NB8672)

- for the **left** connecting circle: K1 =14
- for the **right** connecting circle: K1=24

The word K1 =.. for a connecting circle **surrounding the first circle** is:



- for the **left** connecting circle: K1=15
- for the **right** connecting circle: K1 =25

Note: It depends on the programmed direction of movement (G2 and G3) on the three circles which default value for K1 = is used by the control.

The formats are:

start point from N1 is known

N1	G2/G3	I..	J..	R1=1/2
N2	G2/G3	R..	K1=12/13/14/15 or K1=22/23/24/25	
N3	G2/G3	etc.		

start point from N1 is not known

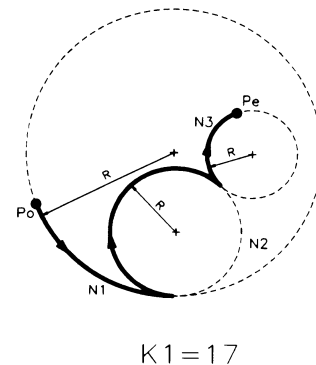
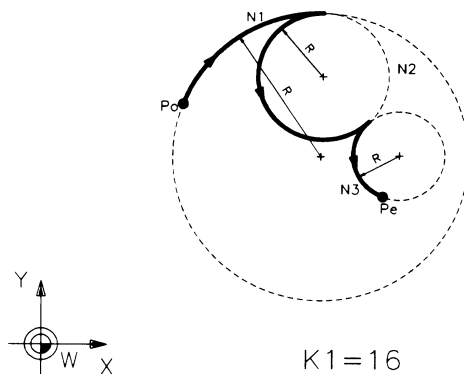
N1	G2/G3	I..	J..	R..	R1=1/2
N2	G2/G3	R..	K1=12/13/14/15 or K1=22/23/24/25		
N3	G2/G3	etc.			

Refer to the corresponding section with continuous movements for the formats of block N1 and N3.

13.4.11 Connecting circle between two circles one inside the other

Two types of connecting circles on the left of the line through the centre points are possible with two circles of which one lies inside the other one. The same two types can be found on the right of the line through the centres.

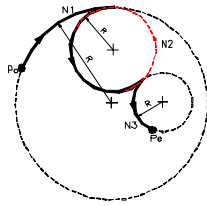
The word K1 =.. for a connecting circle **outside the smaller circle** is:



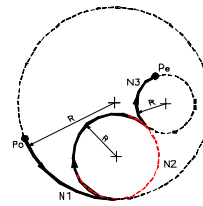
(NB8634)

- for the **left** connecting circle: K1=16
- for the **right** connecting circle: K1 =26

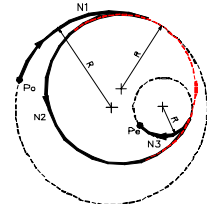
The word K1=.. for a connecting circle **surrounding the smaller circle** is:



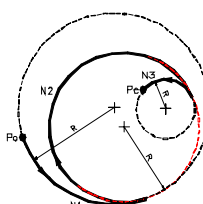
K1=16



K1=17



K1=16



K1=17

- for the **left** connecting circle: K1 =17
- for the **right** connecting circle: K1=27

Note: It depends on the programmed direction of movement (G2 and G3) on the three circles which default value for K1 = is used by the control.

The formats are:

start point from N1 is known

N1	G2/G3	I..	J..	R1=1/2
N2	G2/G3	R..	K1=16/17 or K1=26/27	
N3	G2/G3	etc.		

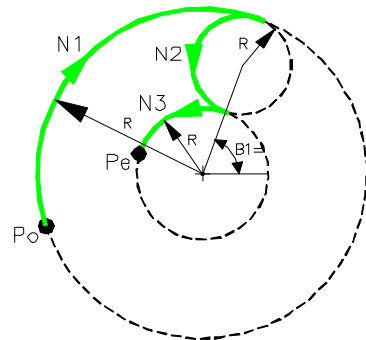
start point from N1 is not known

N1	G2/G3	I..	J..	R..	R1=1/2
N2	G2/G3	R..	K1=16/17 or K1=26/27		
N3	G2/G3	etc.			

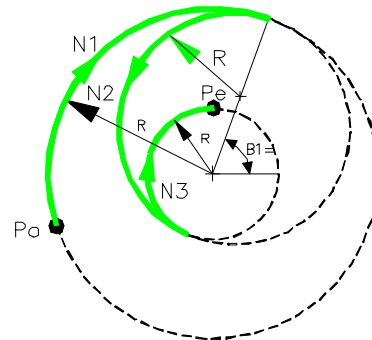
Refer to the corresponding section with continuous movements for the formats of block N1 and N3.

13.4.12 Connecting circle with two concentric circles

If both circles are concentric, the programming is the same as with one circle inside the other one, except that it is also necessary to program the angle (B1=..), which the line through the common centre point and the centre point of the connecting circle makes with the main axis.



K1=16



K1=17

For both cases the following format is available:

start point from N1 and radius of the connecting circle are known

N1	G2/G3	I..	J..	R1=1/2
N2	G2/G3	R..	B1=..	K1=16/17
N3	G2/G3	I..	J..	

start point from N1 and radius of the second circle are known

In this case the radius of the connecting circle is calculated by the control.

N1	G2/G3	I..	J..		
N2	G2/G3	B1=..	K1=16/17		
N3	G2/G3	I..	J..	X..	Y..

Refer to the corresponding section with continuous movements for the formats of block N1 and N3

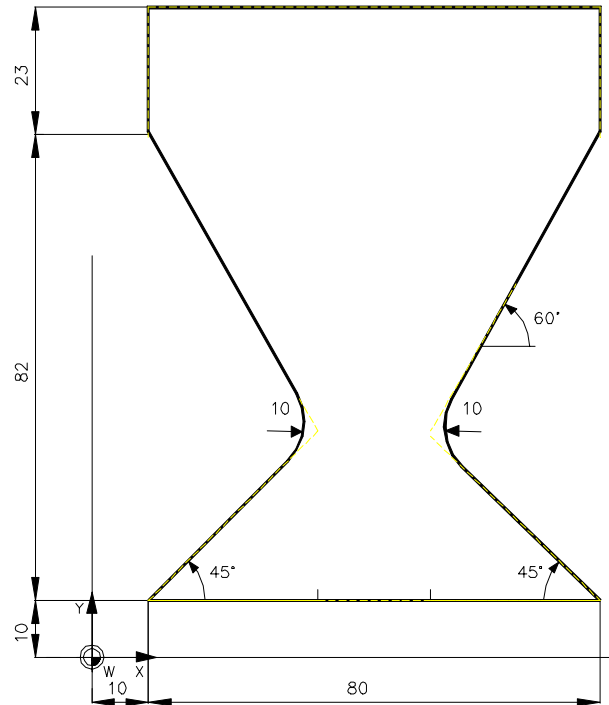
Start point from N1 is known

N1	G2/G3	I..	J..	R..
----	-------	-----	-----	-----

The other blocks from the mentioned cases remain the same.

13.5 Examples

EXAMPLE 1. Calculation of intersection point



```

N64001
N1 G54
N2 S1000 T1 M6 (RADIUS 2 mm)
N3 G0 X0 Y0 Z10 M3
N4 G1 Z-10 F500
N5 G43 Y10
N6 G42
N7 G64
N8 X90
N9 B1=135
N10 G2 R10
N11 G1 X90 Y92 B1=60
N12 Y115
N13 X10
N14 Y92
N15 B1=-60
N16 G2 R10
N17 G1 X10 Y10 B1=45
N18 G40
N19 G63
N20 X0 Y0
N21 G0 Z100
N22 M30

```

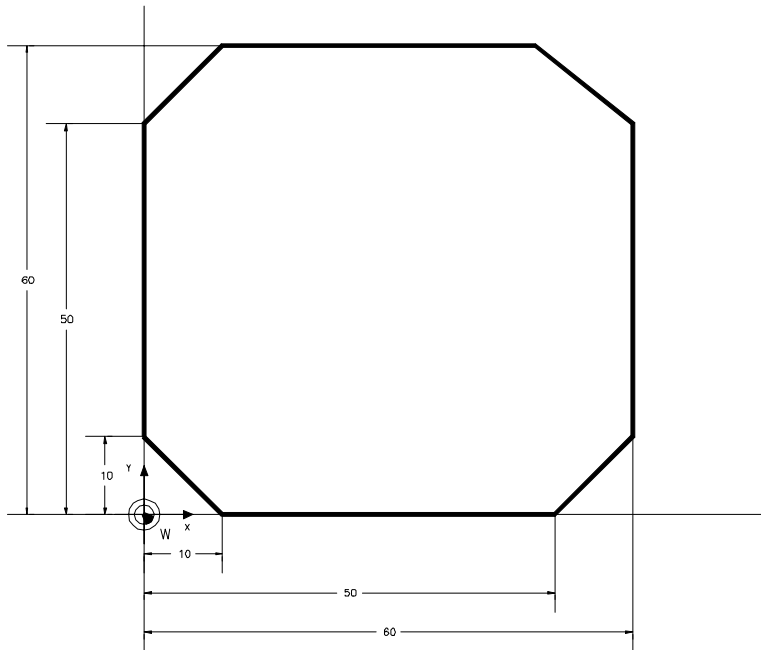
Explanation

N1:	Set the program datum point
N2:	Load tool 1
N3:	Start the spindle and move tool to starting point
N4:	Feed tool to depth
N5:	Move tool T0 the contour
N6:	Set radius compensation RIGHT
N7:	Activate the geometric calculations
N8:	Move tool parallel to X-axis. Y-coordinate can be omitted.
N9:	A linear movement under an angle. The starting point of this line is known, so programming the angle is sufficient
N10:	The rounding between the intersecting lines of N9 and N11
N11:	A linear movement under an angle to an end point.
N12-N14:	Axis parallel movements.
N15-N17:	The linear movements under an angle including the rounding (N16) between these movements.
N18:	Cancel radius compensation
N19:	Cancel geometric calculations
N20:	Move the tool to a point free from the part
N21:	Retract the tool in the tool axis
N22:	End of the program

Notes: To get a graphical presentation of this program in one of the graphic modes of the control, you can add the following lines after N1:

```
N801 G98 X0 Y0 Z0 I100 J130 K-10
N802 G99 X10 Y10 Z0 I80 J105 K-10
```

EXAMPLE 2. Insertion of a chamfer between linear movements



```

N64002
N1 G54
N2 S1000 T1 M6 (RADIUS 2 mm)
N3 G0 X-20 Y-20 Z10 M3
N4 G1 Z-10 F500
N5 G43 Y0
N6 G42
N7 G64
N8 X60
N9 I10
N10 Y60
N11 I10
N12 X0
N13 I10
N14 Y0
N15 I10
N16 X10
N17 G40
N18 G63
N19 Y-20
N20 G0 Z100
N21 M30

```

Explanation

N1 - N7:	Refer to the corresponding lines in the first example.
N8-N10-N12-N14:	Axis parallel movements
N9-N11-N13-N15:	Chamfer of N9 between the linear movements of N8 and N10, chamfer of N11 between N10 and N12 etc.
N16:	Last movement to define the position of the chamfer
N17:	Cancel radius compensation
N18:	Cancel geometric calculations
N19:	Move the tool to a point free from the part
N20:	Retract the tool in the tool axis
N21:	End of the program

Notes:

1. To get a graphical presentation of this program in one of the graphic modes of the control, you can add the following lines after N1:

N801	G98	X-10	Y-10	Z0	I80	J80	K-10
N802	G99	X-5	Y-5	Z0	I70	J70	K-10

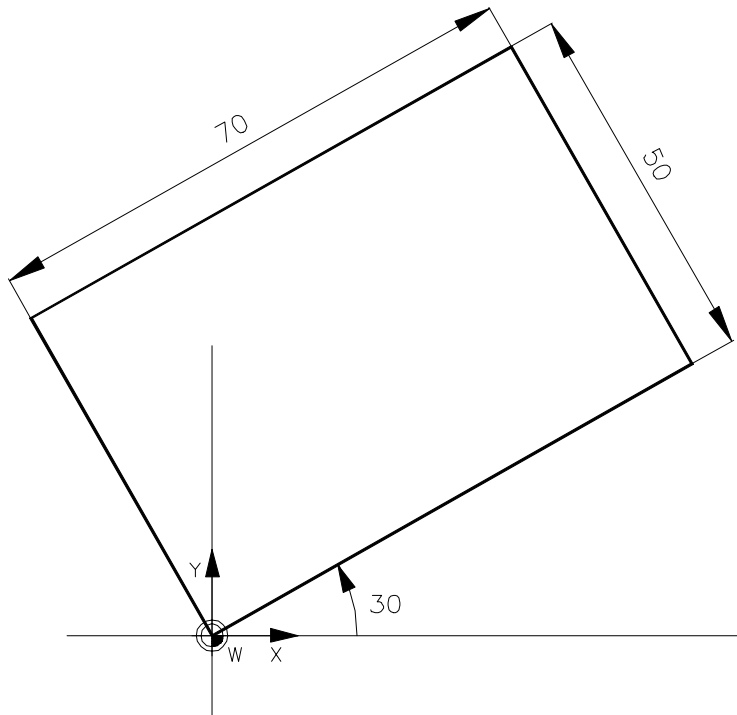
2. If a rounding should be programmed instead of the chamfer a few minor changes have to be made:

```

N9 G3 R10
N10 G1 Y60
N11 G3 R10
N12 G1 etc.

```


EXAMPLE 3. Parallel lines



```

N64003
N1 G54
N2 S1000 T1 M6 (RADIUS 2 mm)
N3 G0 X-10 Y0 Z10 M3
N4 G1 Z-10 F500
N5 G43 X-4.33 Y-2.5
N6 G42
N7 G64
N8 X0 Y0 B1=30 I1=0
N9 X0 Y0 B1=120 I1=70
N10 X0 Y0 B1=-150 I1=50
N11 X0 Y0 B1=-60
N12 G40
N13 G63
N14 X-10 Y-10
N15 Z100
N16 M30

```

Explanation

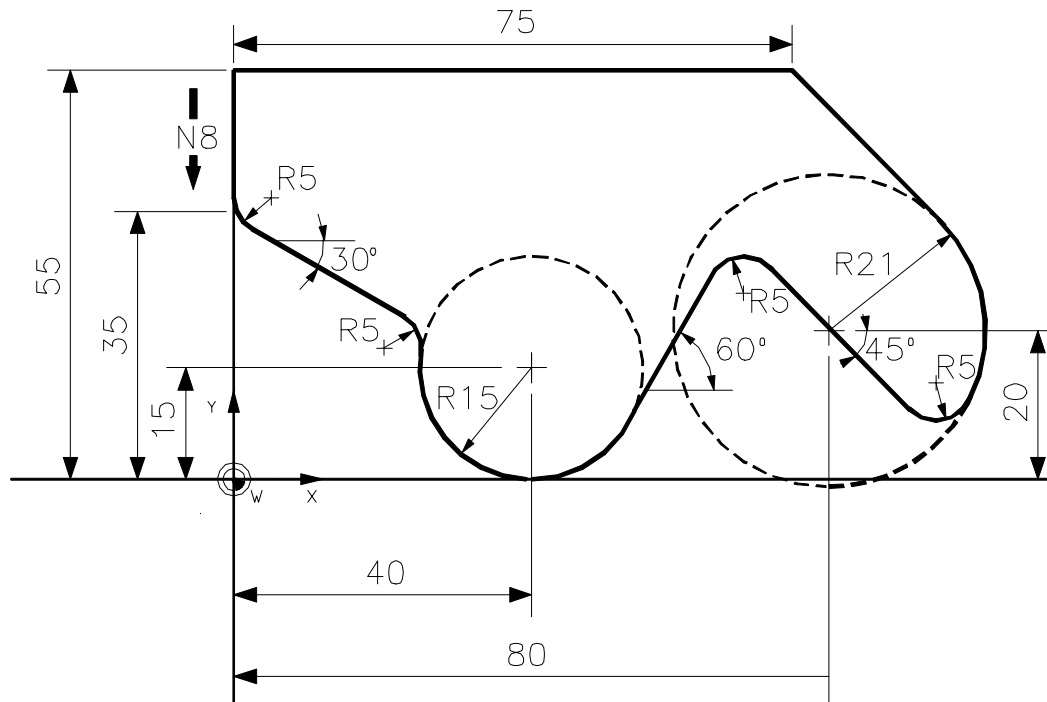
N1 - N7:	Refer to the corresponding lines in the first example.
N8:	Line defined by a support point (X0,Y0) and the angle with the X-axis
N9:	Line defined by a support point (X0, Y0), the angle with the X-axis and at a distance of 70 from the line through the origin.
N10:	Line defined by a support point (X0,Y0), the angle with the X-axis programmed in the direction of movement, and at a distance of 50 from the line through the origin.
N11:	Line defined by the endpoint (X0, Y0) and the angle with the X-axis programmed in the direction of movement.
N12:	Cancel radius compensation
N13:	Cancel geometric calculations
N14:	Move the tool to a point free from the part
N15:	Retract the tool in the tool axis
N16:	End of the program

Note:

To get a graphical presentation of this program in one of the graphic modes of the control, you can add the following lines after N1:

```
N801 G98 X-35 Y-15 Z0 I110 J105 K-10  
N802 G99 X-30 Y-10 Z0 I120 J95 K-10
```

EXAMPLE 4. Line-to-circle and circle-to-line intersection



```

N64004
N1 G54
N2 S1000 T1 M6 (RADIUS 2 mm)
N3 G0 X-5 Y60 Z10 M3
N4 G1 Z-10 F500
N5 G43 X0
N6 G42
N7 G64
N8 B1=-90
N9 G3 R5
N10 G1 X0 Y35 I1=0 B1=-30 J1=2
N11 G2 R5
N12 G3 I40 J15 R15 R1=0
N13 G1 B1=60
N14 G2 R5
N15 G1 X80 Y20 I1=0 B1=-45 J1=2
N16 G3 R5
N17 I80 J20 R21 R1=0
N18 G1 X75 Y55
N19 X-20
N20 G40
N21 G63
N22 G0 Z100 M30

```

Explanation

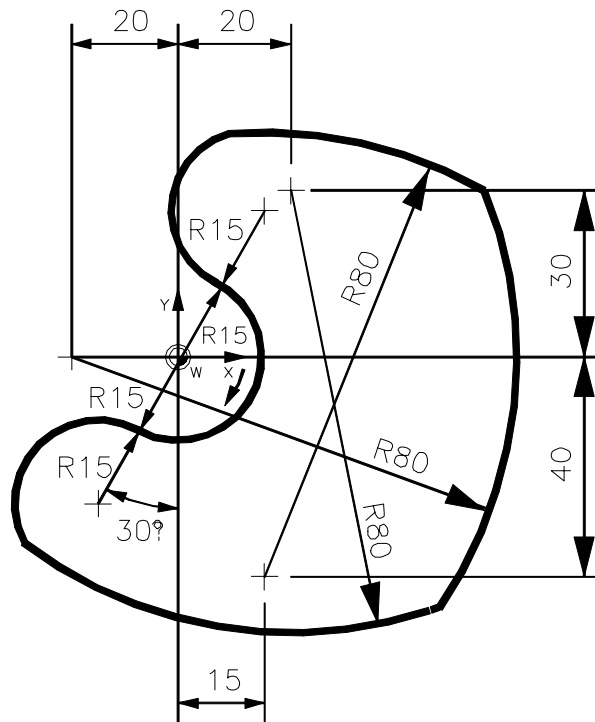
- N1 - N7: Refer to the corresponding lines in the first example.
- N8: Move tool downwards along the Y-axis
- N9: Make the rounding between the linear movements of N8 and N10
- N10: Move tool along the line. The starting point of this line is programmed as a support point, the angle is programmed in the direction of the movement and the right intersection point (J1=2) of the line and the circle of N12 should be used.
- N11: Make the rounding between the linear movement of N10 and the circular movement of N12
- N12: Follow the circle till the point of tangency between the circle and the linear movement of N13
- N13: A linear movement. The point of tangency is the known starting point.
- N14: A rounding between the linear movements of N13 and N15
- N15: A linear movement through the centre of the circle of N17. The centre point is used as a support point of the line. The intersection point in the direction of movement should be used (J1=2)
- N16: Make the rounding between the linear movement of N15 and the circular movement of N17
- N17: Follow the circle till the point of tangency between the circle and the linear movement of N18
- N18: A linear movement to the programmed end point of the line
- N19: Move tool parallel to X-axis, until the tool is free from the part
- N20: Cancel radius compensation
- N21: Cancel geometric calculations
- N22: Retract the tool in the tool axis and end of the program

Note:

To get a graphical presentation of this program in one of the graphic modes of the control, you can add the following lines after N1:

```
N801 G98 X-10 Y-20 Z0 I140 J100 K-10
N802 G99 X0 Y-10 Z0 I120 J80 K-10
```

EXAMPLE 5. Circle-to-circle intersection



```

N64005
N1 G54
N2 S1000 T1 M6 (RADIUS 2 mm)
N3 G0 X0 Y0 Z10 M3
N4 G1 Z-10 F500
N5 G43 X15
N6 G42
N7 G64
N8 G2 R15 R1=0
N9 G3 R15 B3=-120 L3=30 J1=1
N10 I20 J30 R80 J1=1
N11 I-20 J0 R80 J1=1
N12 I15 J-40 R80 J1=1
N13 R15 B3=60 L3=30 R1=0
N14 G2 X15 Y0 R15
N15 G40
N16 G1 X0
N17 G63
N18 G0 Z100 M30

```

Explanation

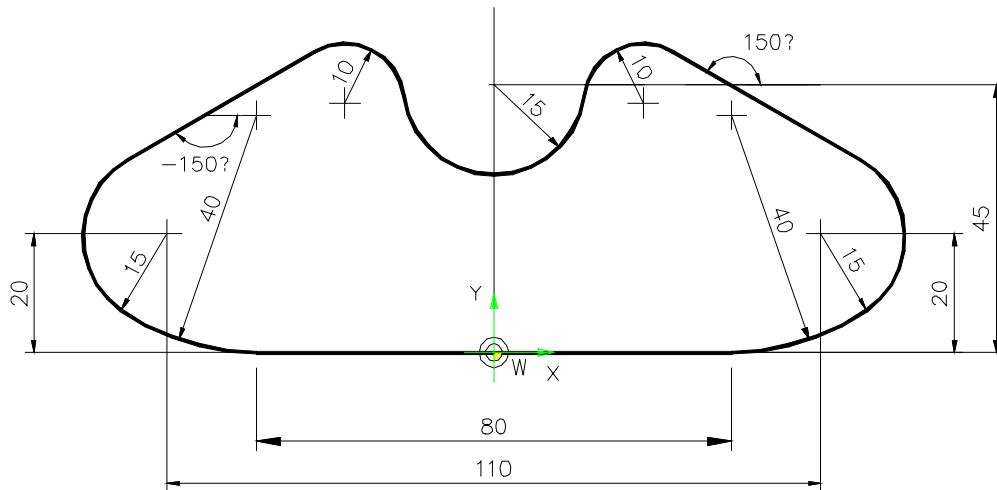
- N1 - N7: Refer to the corresponding lines in the first example.
- N8: Move along the circle till the point of tangency with the next circle. The starting point of this circle is known from the previous blocks.
- N9: Move along the circle defined by its polar centre point coordinates (B3=, L3=..) and the radius. The left intersection point (J1=1) with the circle in N10 should be used.
- N10: Move along the circle defined by its Cartesian centre point coordinates and the radius. The left intersection point (J1=1) with the circle in N11 should be used.
- N11 - N12: The same type of movement as N10.
- N13: Move along the circle till the point of tangency with the circle in N14. The circle is defined by its polar centre point coordinates (B3=, L3=..) and the radius.
- N14: Move along a circle programmed with end point and radius.
- N15: Cancel radius compensation
- N16: Cancel geometric calculations
- N17: Move the tool to a point free from the part
- N18: Retract the tool in the tool axis
- N19: End of the program

Notes:

To get a graphical presentation of this program in one of the graphic modes of the control, you can add the following lines after N1:

```
N801 G98 X-45 Y-60 Z0 I140 J140 K-10
N802 G99 X-40 Y-60 Z0 I125 J120 K-10
```

EXAMPLE 6. Connecting circle between line and circle



```

N64006
N1 G54
N2 S1000 T1 M6 (RADIUS 2 mm)
N3 G0 X0 Y-20 Z10 M3
N4 G1 Z-10 F500
N5 G43 Y0
N6 G42
N7 G64
N8 B1=0
N9 G3 R40
N10 G3 I55 J20 R15 R1=0
N11 G1 B1=150
N12 G3 R10
N13 G2 I0 J45 R15
N14 G3 R10
N15 G1 B1=-150 R1=0
N16 G3 I-55 J20 R15
N17 G3 R40
N18 G1 X0 Y0 B1=0
N19 G40
N20 G63
N21 G0 Z100
N22 M30

```

Explanation

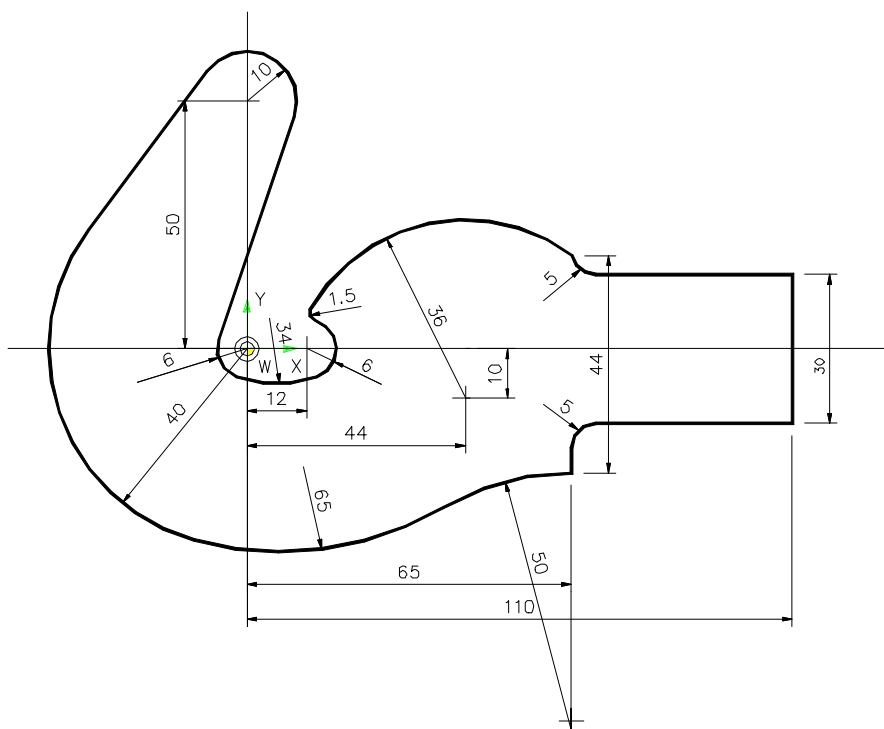
N1 - N7:	Refer to the corresponding lines in the first example.
N8:	Move along the X-axis. The starting point of the line is known from the previous blocks.
N9:	A surrounding connecting circle between the line of N8 and the circle of N10. The line does not meet the circle.
N10:	A circular movement till the point of tangency with the linear movement of N11. The circle is programmed with the Cartesian centre point coordinates and the radius.
N11:	A linear movement. The starting point of the line is the point of tangency with N10. The line does not meet the circle of N13.
N12:	A connecting circle between the line of N11 and the circle of N13.
N13:	A circular movement from the point of tangency with the circle of N12 till the point of tangency with the circle of N14.
N14:	Similar to N12.
N15:	A linear movement to the point of tangency with the circle from N16.
N16:	A circular movement till the point of tangency with the surrounding connecting circle of N17
N17:	The surrounding connecting circle.
N18:	A linear movement along the X-axis to three programmed end point
N19:	Cancel radius compensation
N20:	Cancel geometric calculations
N21:	Retract the tool in the tool axis
N22:	End of the program

Note:

To get a graphical presentation of this program in one of the graphic modes of the control, you can add the following lines after N1:

```
N801 G98 X-90 Y-15 Z0 I180 J90 K-10
N802 G99 X-70 Y-5 Z0 I140 J60 K-10
```


Programming manual milling



671

Explanation

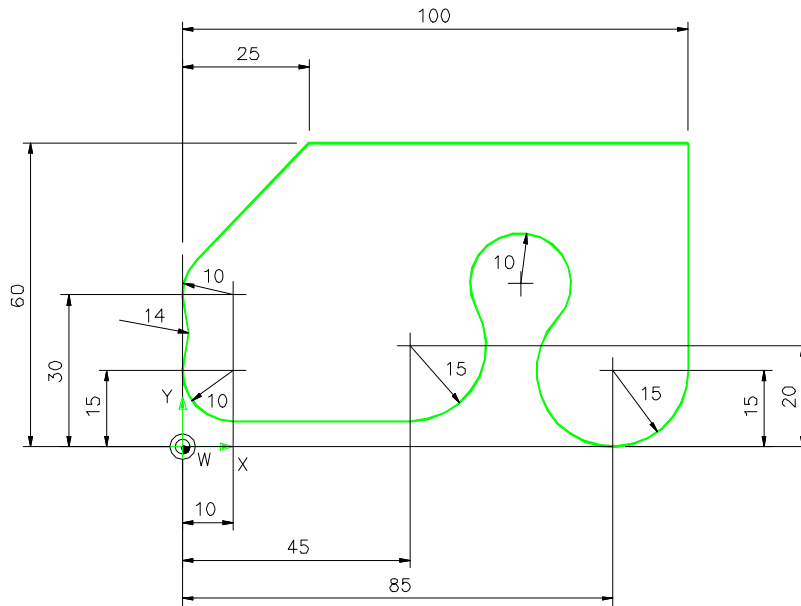
N1 - N7:	Refer to the corresponding lines in the first example. The radius compensation is set to be LEFT.
N8:	Move parallel to the X-axis. The starting point is known from the previous blocks.
N9:	A rounding between the linear movements of N8 and N10
N10:	A downwards-linear movement parallel to the Y-axis till the left intersection point of the circle from N11
N11:	A circular movement to the starting point of the connecting circle of N12
N12:	Connecting circle outside the circle of N11 and surrounding the circle of N13
N13:	Circular movement till the point of tangency with the line of the next block
N14:	Common tangent line between the circles of N13 and N15
N15:	Circular movement to the point of tangency with the line of the next block
N16:	Common tangent line between the circles of N15 and N17
N17:	Circular movement to the point of tangency with the connecting circle of the next block
N18:	Connecting circle which surrounds the circles from N17 and N19
N19:	Circular movement to the point of tangency with the rounding of the next block. The circle intersects the circle of block N21
N20:	The rounding between the two intersecting circles of N19 and N21
N21:	Circular movement to the point of tangency with the line of the next block.
N22:	Linear movement programmed with its end point
N23:	A downwards linear movement parallel to the Y-axis
N24:	A rounding between the linear movements of N23 and N25
N25:	Linear movement parallel to the X-axis programmed with end point and angle
N26:	Downward linear movement parallel to Y-axis
N27:	Cancel radius compensation
N28:	Cancel geometric calculations
N29:	Move the tool to a point free from the part
N30:	Retract the tool in the tool axis and end of the program

Note:

To get a graphical presentation of this program in one of the graphic modes of the control, you can add the following lines after N1:

```
N801 G98 X-50 Y-50 Z0 I170 J120 K-10
N802 G99 X-45 Y-45 Z0 I160 J110 K-10
```

Example 8. Connecting circle with greater arc selection



```

N64008
N1 G54
N2 S1000 T1 M6 (RADIUS 2 mm)
N3 G0 X115 Y60 Z10 M3
N4 G1 Z-10 F500
N5 G43 X100
N6 G41
N7 G64
N8 R1=0
N9 G2 I85 J15 R15
N10 G3 R10 K1=22
N11 G2 I45 J20 R15 R1=0
N12 G1 R1=0
N13 G2 I10 J15 R10
N14 G3 R14
N15 G2 I10 J30 R10 R1=0
N16 G1 X25 Y60
N17 X115
N18 G40
N19 G63
N20 Z100
N21 M30

```

Explanation

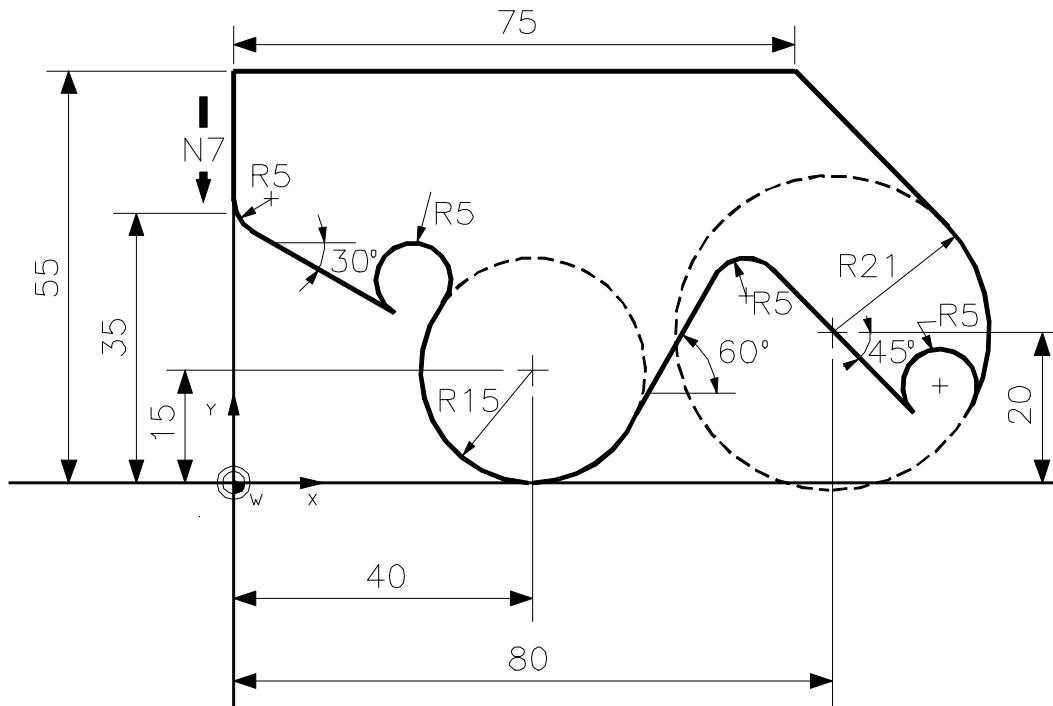
N1 - N7:	Refer to the corresponding lines in the first example. The radius compensation is set to be LEFT.
N8:	Linear movement. The line is tangent to the circle of N9. The starting point of the line is known from the previous blocks.
N9:	Circular movement to the point of tangency with the connecting circle of the next block
N10:	Connecting circle between the circles of N9 and N11. The greater arc is required. In this case the indicator K1=22 should be programmed. If this indicator is omitted the shorter arc (K1 =12) is automatically chosen by the control.
N11:	Circular movement to the point of tangency with the common tangent line of N12.
N12:	Common tangent line between the circles of N11 and N13.
N13:	Circular movement to the point of tangency with the connecting circle of the next block
N14:	Connecting circle on the outside of the circles from N13 and N15
N15:	Circular movement to the point of tangency with the line of N16.
N16:	Linear movement to the programmed end point.
N17:	Linear movement parallel to the X-axis
N18:	Cancel radius compensation
N19:	Cancel geometric calculations
N20:	Retract the tool in the tool axis
N21:	End of the program

Note:

To get a graphical presentation of this program in one of the graphic modes of the control, you can add the following lines after N1:

```
N801 G98 X-10 Y-10 Z0 I120 J80 K-10
N802 G99 X-5 Y-5 Z0 I110 J70 K-10
```

Example 9. Non-continuous movement between line and circle



```

N64009
N1 G54
N2 S1000 T1 M6 (RADIUS 2 mm)
N3 G0 X-5 Y60 Z10 M3
N4 G1 Z-10 F500
N5 G43 X0
N6 G42
N7 G64
N8 B1=-90
N9 G3 R5
N10 G1 X0 Y35 I1=0 B1=-30 J1=2
N11 G2 R5 K1=4
N12 G3 I40 J15 R15 R1=0
N13 G1 B1=60
N14 G2 R5
N15 G1 X80 Y20 I1=0 B1=-45 J1=2
N16 G2 R5 K1=4
N17 G3 I80 J20 R21 R1=0
N18 G1 X75 Y55
N19 X-20
N20 G40
N21 G63
N22 G0 Z100 M30

```

Explanation

Compare this program with the one of example 4. The differences are:

N11: The direction of rotation on the circle is changed and the indicator K1= 4 programmed to make the required circular movement.

N16: The same changes as in block N11.

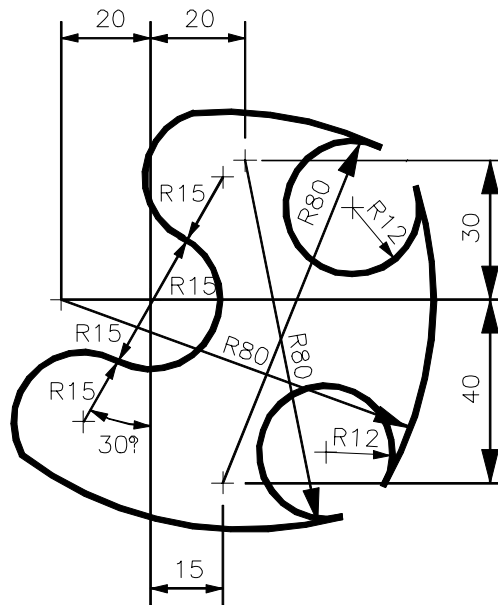
Note:

To get a graphical presentation of this program in one of the graphic modes of the control, you can add the following lines after N1:

N801 G98 X-10 Y-20 Z0 I140 J100 K-10

N802 G99 X0 Y-10 Z0 I120 J80 K-10

Example 10. Non-continuous movement between two circles



```

N64010
N1 G54
N2 S1000 T1 M6 (RADIUS 2 mm)
N3 G0 X0 Y0 Z10 M3
N4 G1 Z-10 F500
N5 G43 X15
N6 G42
N7 G64
N8 G2 R15 R1=0
N9 G3 R15 B3=-120 L3=30 J1=1
N10 I20 J30 R80 J1=1
N11 G2 R12 K1=4
N12 G3 I-20 J0 R80 J1=1
N13 G2 R12 K1=4
N14 G3 I15 J-40 R80 J1=1
N15 R15 B3=60 L3=30 R1=0
N16 G2 X15 Y0 R15
N17 G40
N18 G1 X0
N19 G63
N20 G0 Z100
N21 M30

```

Explanation

Compare this program with the one of example 5. The differences are the roundings of the blocks N11 and N13. The indicator K1 =4 is programmed to make the required circular movement between the intersecting circles.

Notes:

To get a graphical presentation of this program in one of the graphic modes of the control, you can add the following lines after N1:

```

N801 G98 X-45 Y-60 Z0 I140 J140 K-10
N802 G99 X-40 Y-60 Z0 I125 J120 K-10

```


14. Appendix

14.1 Tilting of the operating plane

14.1.1 Introduction

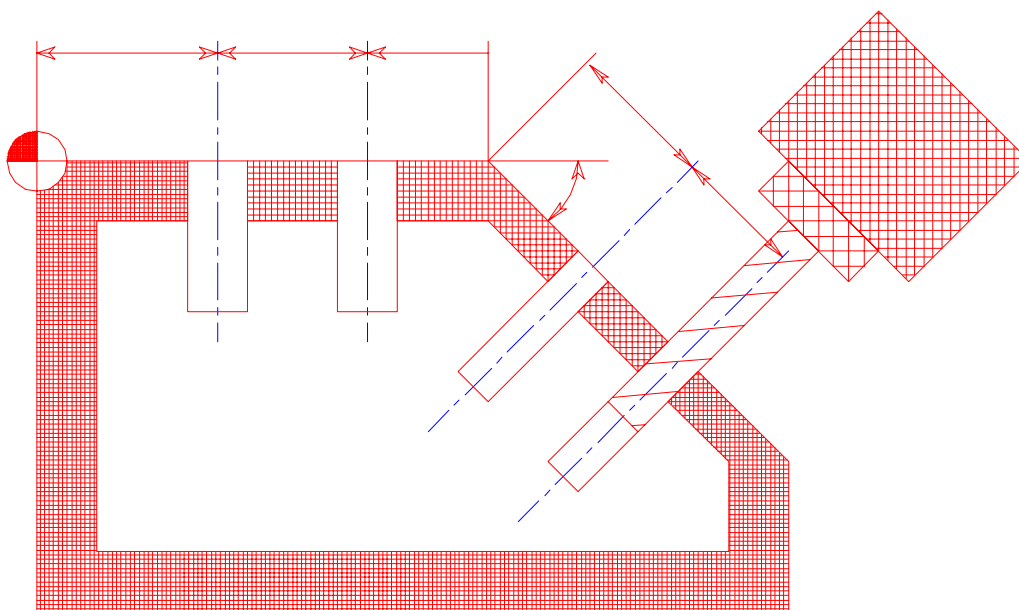
The control supports the tilting of operating planes on tool machines with tilting faces or tilting tables. Please consult your user handbook.

Typical applications, for example, are oblique drilling or contours, which lie obliquely in the operating area. In this way, the operating plane is always tilted about an active null point. Normally, the operation is programmed in a principal plane, e.g. X/Y plane; however, it is executed in the plane, which was tilted to the principal plane.

Consult the description of the G7 function for the programming of the freely programmable operating plane.

The tilting of the operating plane is defined and implemented using the G7 function. The G7 function is made up of two components:

- Definition of new operating planes, rotation of the coordinate system.
- In the event that it is programmed, tilt the tool vertically to the defined operating plane.



An operation on an oblique workpiece plane is programmed in local coordinates. In this way, the local X and Y coordinates lie in the oblique plane and the Z coordinate is fixed vertically in the plane. The machine recognizes the link between the local coordinates and the true machine axes and calculates this. The control calculates the tool correction factor.

Millplus distinguishes two machine types during tilting of the operating plane:

- 1) Machine with a tilting table
The location of the transformed machine axis is not changed with reference to the coordinates system fixed in the machine. If you rotate your table, and thus the workpiece, through 90°, for example, the coordinate system is not rotated through 90° with it. If, in the operating mode "Manual operation", you press the axis direction key Z+, the tool travels in the direction Z+.
- 2) Machine with a tilting face
The location of the tilted (transformed) machine axis is changed with reference to the coordinates system fixed in the machine:
If you rotate the tilting face of your machine and hence the tool, e.g. in the B axis about +90°, the coordinates system is rotated with it. If, in the operating mode "Manual operation", you press the axis direction key Z+, the tool travels in the direction Z+ and X+ of the coordinates system fixed in the machine.

Using the G7 function you define the location of the operating plane by the input of tilt angles. The angles entered describe the angular components of a space vector.

If you program the angular components of the space vector, the control automatically calculates the angular position of the tilt axes. MillPlus calculates the location of the space vector and thus the location of the spindle axis, by means of rotation about the coordinates system fixed in the machine. The sequence of rotations for the calculation of the space vector is fixed: MillPlus turns the A axis first, next the B axis and finally the C axis.

The G7 function is effective from the start of its definition in the program.

MillPlus can only position-controlled axes automatically.

In the G7 definition, you can, in addition, input a safety distance to the tilt angles, with which the tilt axes are positioned.

Use only pre-set tools (full tool length in the tool table).

During the tilting process the position of the tooltip opposite the workpiece remains virtually unchanged (depending on the type of movement L1=).

MillPlus implements the tilting process using the power traverse.

14.1.2 Machine types

Milling machines with four or five axes can be used for the oblique machining of a workpiece.

Depending on the plane, which is tilted, other types of machine are needed for the working. At least two rotary axes and three linear axes are needed, in order to reach all sides and planes (except the under surface) without the need for remounting.

The possible types of machine are:

90° tilting face and turntable

The tilting face can be in two states. The upper and reverse sides can be worked by means of the tilting face. The four side surfaces can be worked using the turntable (C axis). The machine is only suited to all oblique operating planes if the tilting face can also be set in the oblique position manually.

Double turntable

The tables (A and C axes) are stacked. In this way, all sides and oblique operating planes can be worked.

Double turntable and 45° tilting face

The tables (A and C axes) are stacked. The A-axis is limited in its angular movement. In conjunction with the two tilting face states all sides and oblique operating planes can be worked.

45° double turntable

The tables (B and C axes) are stacked. The B axis is fixed in this way at an angle of 45°. All sides and oblique operating planes can be worked.

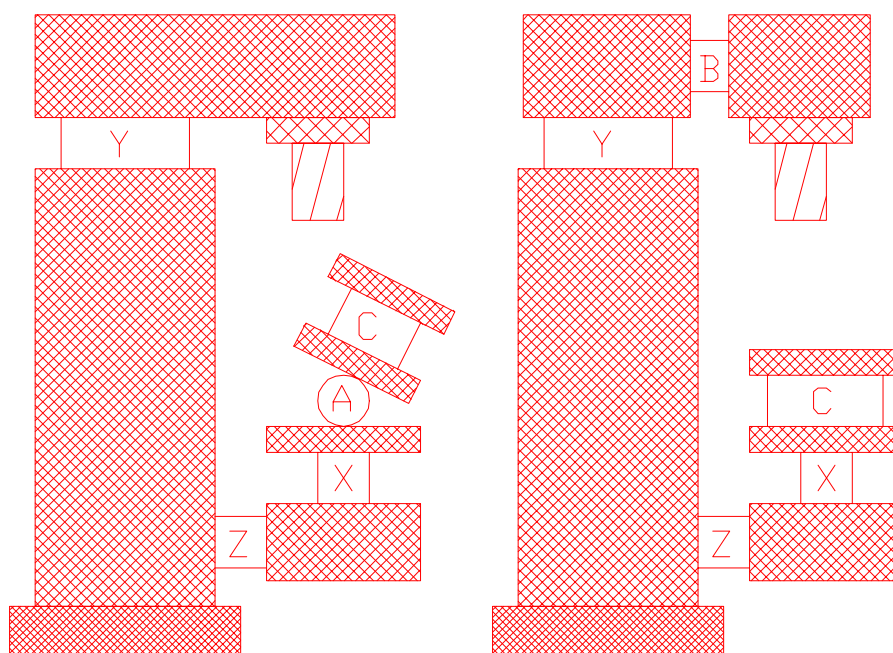
Turntable and rotating face

The face (B axis) can be freely positioned. In conjunction with the table (C axis) all sides and oblique operating planes can be worked.

Turntable and 45° rotating face

The face (B axis) is set at an angle of 45°. In conjunction with the table (C axis) all sides and oblique operating planes can be worked.

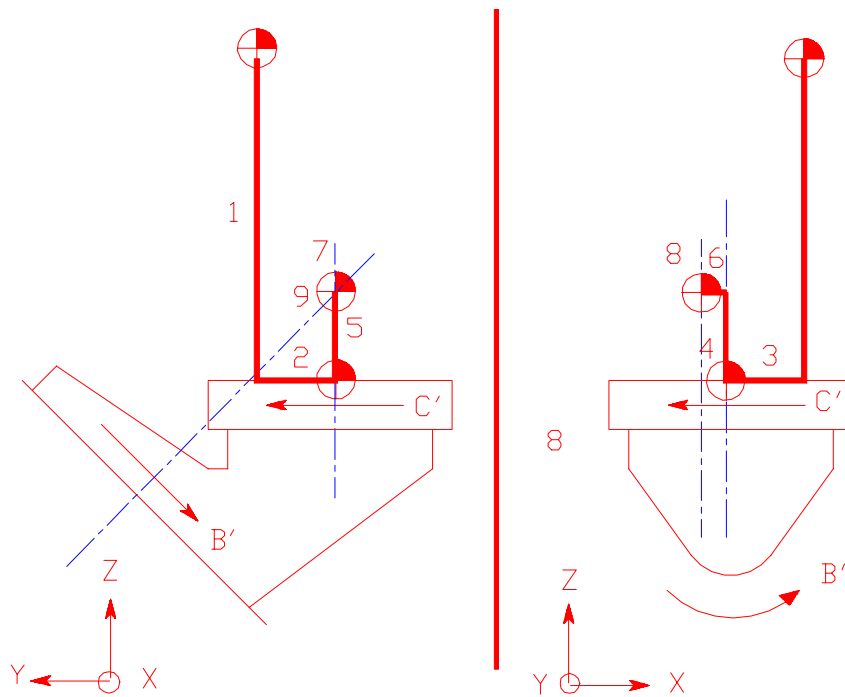
Outline of the most suitable machine types for use with oblique operating planes.



14.1.3 Kinematics model

The control needs a kinematics model of the machine in order to reset the programmed local coordinates in the oblique plane to the movements of the machine axes. A kinematics model describes the "construction" of the axes and the exact position of the different turning points on the rotary axes. As an example, a kinematics model of the DMU 50 V machine is shown. The kinematics model is made up of a chain stretching from the workpiece to the machine frame. It is not necessary to describe the chain from the workpiece to the machine frame, because it includes no rotary axes.

Kinematics model for the DMU 50 V



Explanation of the drawing:

- 1,2,3 three elements in the X, Y, and Z directions in order to fix the (absolute) centre position of the workpiece table with reference to the marker positions.
- 4 element for definition of the C axis.
It is only necessary to describe the rotating axis of a rotary axis, not the centre point.
- 5,6 two elements in order to obtain the rotating axis of the second (incremental) rotary axis.
- 7 element for definition of the (incremental) direction of the second rotating axis. This direction is -45° in the A axis (all around the X axis).
- 8 element for definition of the B axis.
- 9 element in order to raise the -45° tilt (Element 7) again. In this way, the kinematics chain ends without rotation.

The kinematics model is entered by means of the machine settings MC600 to MC699.

A kinematics chain defines, by means of displacements and tilting, the way in which the rotary axes lie with respect to one another. Every displacement or tilting is determined as an element of the kinematics chain in three machine settings. Thus, a total of 25 elements of the kinematics chain can be determined. All rotary axes and positioning axes, which are present, should be described.

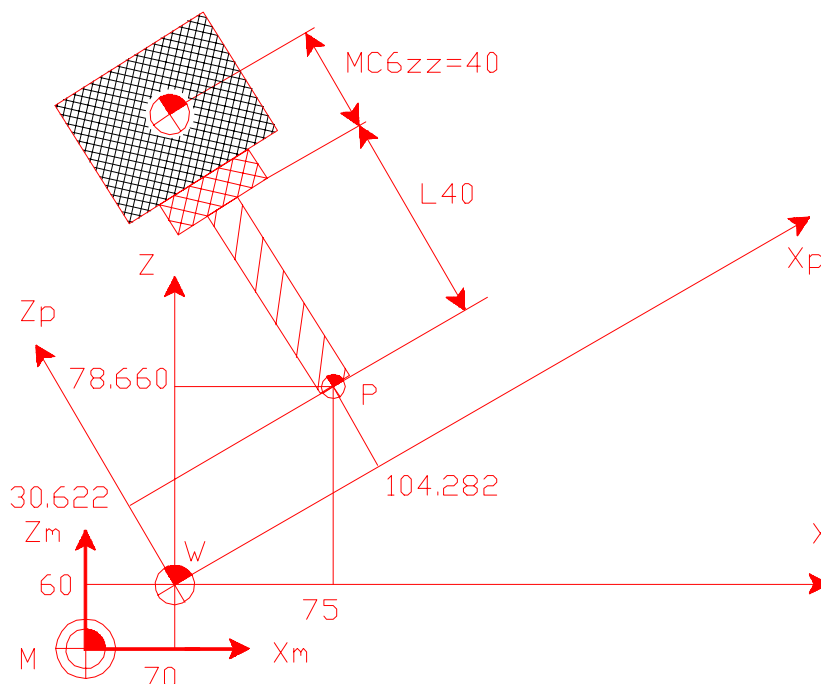
The axes are used along the local coordinates within the tilted G7 plane. E.G. Jogging skip of the Z-axis moves the tool vertically in the plane. All true linear machine axes can move in this way.

By means of a soft key, operation can be switched to the use of the true machine axes. The display then changes to show the true machine axes.

The user keys and the hand wheels for the linear axes can be assigned, according to choice, to the G7 plane or to the machine axes. The display is then implemented also in the G7 or machine axes plane. A new soft key in the soft key group <Step/continue> is used to implement the choice between G7 level or machine axes. For this purpose, this new soft key offers a "pause" option between the choices of jogging skip <advance> and <continue> soft key.

14.1.4.2 Display

If G7 is active, a yellow icon is displayed on the screen behind the tool number. By means of a small "p" on the right next to the "axis characters" an indication is given as to whether the display relates to the tilted operating planes or to the machine coordinates. The operating status has been enhanced with the current state of the programmed G7 space angle.



It is possible, using a new soft key in the "Jog operation type" soft key group, to change the display at the same time as the jog direction. If the position is displayed in machine coordinates, the position of the true tooltip is shown. See the next screen:

The position display on the screen can change between the position in the G7 plane (X_p , Z_p) or in machine coordinates (X , Z).

Both are based on the active null point G52 + G54 + G92/G93.

14.1.4.3 Axis selection/position axis

An axis, which is not regulated, must be set to the correct position by hand. However, either before or after this, the oblique setting of the tool must also be entered by means of G7. Otherwise it will not be checked.

Comment: The expected position of the rotary axes is set parametrically in G7 using $n7=<\text{parameter number}>$. An axis selection or a positioning axis can be set manually using this information.

The axis selection or the positioning axis should also be followed in the kinematics model.

14.1.4.4 Reference point

If the reference point is approached during G7, the rotary axes remain in their reference position following the approach. The G7 plane is deactivated and the G17 plane is activated.

After running up the machine, but before approaching the reference point, the G7 plane is still active. After < reset CNC > the G7 plane is deactivated.

14.1.4.5 Intervention

If the G7 movement is interrupted, the exact position of the tooltip is displayed on the screen. Following interruption, the axes can be used in manual operation [mode].

Comment: Manual operation of the axis corresponds with the interrupted G7 plane (or, according to choice, the machine axes).

Following <Start> a move in position back to the interrupted point is affected. At the same time the axes run with positional logic corresponding with the G7 plane. Concurrently, the rotary axes rotate to the initial position.

14.1.5 Error messages

P306 plane not clearly defined

The G7 plane is defined using a mix of absolute angles ($A5=$, $B5=$, $C5=$) and incremental angles ($A6=$, $B6=$, $C6=$).

Resolution: Use only absolute or incremental angles. If necessary, several G7 definitions with incremental angles can be defined, one behind the other.

P307 program plane not attainable

The G7 oblique setting defined cannot be attained, on account of the limited range of the rotary axes.

Resolution: Machines with a tilting face should tilt the face (by means of the M function) from the instantaneous position (horizontal or vertical) following the other position

O256 Machine type not recognized

The kinematics model in MC600 to MC699 is defining a type of machine, which is not supported for the oblique operating plane (G7). Only machine types with the following sequence of rotary axes, as viewed from the workpiece to the tool, are supported:

- A C
- C A
- C B
- C A fixed B -A fixed
(A fixed is a fixed tilt in the direction of the A axis, as, for example, the DMU50V has with -45°)
- C
- Axis change variants (C becomes B, and B becomes C) are also possible.

Resolution: The kinematics model should be entered in detail, with at least a description of those rotary axes present. The control must be run up once more.

14.1.6 Machine Constants

MC312 free operating plane (0=off, 1=on)

Activates the free operating plane. The G7 function can be programmed.

MC600 - MC699

There are 100 new machine settings (MC600 - MC699) for the description of the kinematics model. The model is described using a maximum of 25 elements, in which each element is described by means of four machine settings.

The following machine settings are used:

MC600 Kinematics chain (0=end, 1=tool, 2=workpiece)

MC601 Element (0,1=X, 2=Y, 3=Z, 4=A, 5=B, 6=C)

MC602 Element type (0=incremental, 1=absolute, 2=programmable)

MC603 Element insertion [m/m degrees]

MC604, 608, 612, 616, 620, ..., 696 as MC600

MC605, 609, 613, 617, 621, ..., 697 as MC601

MC606, 610, 614, 618, 622, ..., 698 as MC602

MC607, 611, 615, 619, 623, ..., 699 as MC603

MC755 Free operating plane: rotation (0=coordinates cross, 1=axes)

If the desired rotation of the operating plane corresponds with the rotation of a rotary axis, the control has the choice between rotation using the relevant rotary axis or rotations using the coordinates cross. This choice is made with MC755.

E.G on a machine with a (true) C axis the program instruction G7 C5=30 and MC755=0 produces a rotation of the coordinates cross over -30° and MC755=1 a rotation of the C axis over 30°.

14.2 Look Ahead Feed (LAF) function

14.2.1 Introduction

The Look Ahead Feed function is used to carry out a precalculation on the programmed toolpath, while taking account of the dynamics of all axes involved. The toolpath speed is then adjusted to achieve the highest contour accuracy at the highest possible speed. However, the programmed feed is never exceeded.

Taking account of the programmed feed and the actual feed override setting; special high-performance algorithms ensure a homogenous feed for fast finishing times.

The execution speed of CAD-generated programs is substantially increased.

The user need not look at anything else when working with Look Ahead Feed.
This function cannot be influenced.

Only the G28 function was changed. The addresses for feed limitation were cancelled (see G27/628, starting from V320). Existing programs need not be adapted; they can be run as usual. These functions are ignored during machining operations. The machining operation may, however, continue.

During Look Ahead Feed the end point and centre point of a circle should match within 64 µm. In this case, the centre point is automatically corrected. As opposed to V310, there is no "compensation movement" at the end point. An error message is given if the end point and centre point do not match within 64 µm. This also applies to helix interpolation.

14.2.2 Detailed specification

1) Types of interpolation

The LAF function is active during:

- G0 Rapid traverse, including infeed movements
- G1 Feed movement
- G2, G3 Circle, including helix interpolation

The LAF function is inactive during:

- G6 Splines
- G74 Positioning movements
- G84 Threading
- G145, G45, G46 Measuring movement
- G182 Cylinder jacket interpolation with all permitted movements
- PLC-controlled axis movements (Home position)
- Auxiliary axis movements

Circular interpolation

- The circular accuracy achieved with LAF at higher speeds is higher than that with V3.10. This is true of circles made with G2/G3 and with cycles.

2) Previous contour accuracy functions (G28 function)

The following G28 programming functions are no longer active:

- I3=2 G1, G2, G3 with corner release distance (MC136)
- I3=3 G1 with programmable contour accuracy, (MC137) or I7=(0-10000)[μm]
- I4=2 G0 with corner release distance (MC136)
- I6=1 G2, G3 with feed limitation (MC135)

These functions are ignored during machining. The machining operation may, however, continue.

4) New error messages

P300 LAF: End point not on circle

Circle end point deviation exceeds 64μm.

The following applies additionally to a cylinder: $100 \text{ mm} < R\text{-cylinder} < 10 \text{ m}$

Remedy: The end point should be defined more accurately.

P301 LAF: SW limit switch approached

The programmed path will go beyond the limit switches or outside + or - 100 m. In the case of straight lines this error is generated at the beginning of the wrongly programmed block. In the case of circles, it depends on the circular form and speed.

Remedy: The path should be defined within the possible range.

P302 No interpolation axis

The wrong axes have been defined for the selected type of interpolation:

- No two main axes for circular interpolation
- No rotary axis for cylinder jacket interpolation

Remedy: The missing axis should be defined.

Example

```

N14803 (TEST PROGRAM)
N1 (MAX. RUNTIME 24 MINUTES)
N1 (LAST BLOCK NUMBER N120666)
N6 G90
N12 G40
N14 M9
N16 T2 M67
N18 (TOLERANCE 0.01)
N20 (SPHERICAL CUTTER D 8 mm, OFFSET 0.0 )
N24 F15000 S16000 M3
N26 G0 X230 Y-14.8 Z10
N28 G0 Z42.5
N30 G1 Z32.5
N32 X210.16
.
N120664 G0 Z100
N120666 M30

```

14.3 Graphical support

The CNC PILOT provides the following types of graphical support:

- synchron graphic

The actual tool movements on the machine are simultaneously displayed on the control.

- Graphical simulation:

The visualization of the programmed tool movements on the display of the control.

- Graphical programming support

To get a graphical presentation on the display of the control the proper display mode must be chosen on the control, but also the partprogram should be enlarged with some special G-functions:

- G195 for defining the 3D window, thus a bounded area on the display of the control in which the graphical simulation takes place;
- G196 to G199 for defining the inner and outer contour of the blank part and the machine parts.

14.3.1 Synchron graphic

With synchron graphic a 3D view of the cutter path (= path of the tooltip) is displayed on the control simultaneously with the actual tool movements on the machine tool. Feed movements and rapid traverse movements are displayed with different colours.

The graphic window defined with the G195 function is used to set the borders on the screen. There is no scale available to make any measurement.

The synchron graphics is available in the mode OPERATE of the control.

14.3.2 Graphical simulation

Graphical simulations allow programmed tool movements to be visualized on the display of the control. The visualization includes:

- A wire plot graphic showing the contour, the tool movements, the cycle end points, the machine parts and the tool shape.
- An erase graphic showing the machining of the workpiece and if required indicating a collision between tool and workpiece with rapid traverse movements or machine parts,
- A workpiece presentation showing the machined workpiece and the intersections.

Graphical simulations of a partprogram are available in the mode OPERATE of the control.

Note: Rotary axes are not simulated.

WIRE PLOT GRAPHIC

In wire plot graphic the outer and inner contour of the blank, the machine parts and the tool shape are shown. The tool movements along the part are generated in either SINGLE BLOCK or AUTOMATIC mode. The functions BLOCK DELETE and OPTIONAL STOP can also be executed during the execution of the program.

The axis cross can be shown to measure the part dimensions and the position of the tool.

A mill is represented as a circle. Its size is picked up from the tool memory.

In all plot modes except the 2.5 D plot, a picture of the tool holder and the tool defined with the G-word from the tool memory can be displayed in a separate window.

The programmed model of the blank, the machine parts, the tool shape picture, the programmed contour, the feed and the rapid traverse movements are all displayed in different colours.

Collisions during rapid traverse movements between the tool and the workpiece or collisions with machine parts can be seen.

The end contour of a fixed milling cycle is shown as a programmed contour.

With the universal pocket cycle the programmed contour of the pocket and the islands are shown when the function G200 is processed. Then the macros are generated and the pocket milling simulated.

A zoom function is available to expand or reduce the window for viewing the part

2D plot

With a 2D plot both the contour and the cutter path in the active main plane can be displayed. In this way the programmed contour and the cutter path along that contour can be checked.

2.5D plot

With a 2.5D plot the tool movements in the active main plane and in the projections are shown.

With a machine constant setting either European or American projection can be chosen.

In a small extra window a 3D plot of the part can be shown.

In this plot mode the contour and the cutter path along that contour can be checked, but also the movements in the tool axis.

In the 2D and 2.5D plot mode a cross hair function is available to check the part dimensions. The position of the cross is shown in the top window.

3D plot and Perspective view

With the 3D plot and the perspective view the tool movements are shown in a spatial representation of the workpiece.

With the 3D plot parallel lines remain parallel and with the perspective view they meet at a particular horizon.

In both plot modes the part can be rotated about the three main axes to show the part under different angles.

ERASE GRAPHIC

With erase graphics a solid model of the part is displayed in either a 2D or a 2.5D view. In this model the machining of the part is shown and if required a collision is detected between the part and the tool during rapid traverse movements or with the machine parts, if programmed in the graphical model. Different colours are also used for different depths of operation.

A zoom function is available to expand or reduce the window for viewing the part

An axis cross is not available. Measurements on the display can be executed in either the wire plot graphic or the workpiece presentation.

COLLISION DETECTION WITH ERASE GRAPHIC

The programmed contours of the machine parts are not used for collision detection, but instead the smallest possible rectangle is calculated which encloses the whole of the machine part(s). When a tool is detected crossing into this rectangle a collision warning is generated.

The softkey 'COLL DETECT' must be activated for a collision message to be generated during a graphical simulation. If this softkey is not activated, the graphic tool will just draw over the machine part.

WORKPIECE PRESENTATION

With workpiece presentation the total program is executed by the control in a fast way, then the part and its intersections are displayed. The user can select the intersection and move the cross hair to the point he wants to check. The part is then displayed in all three intersections.

14.3.3 Graphical programming support

With programming via the control, thus entering a program via the control, three types of support are available:

a. Static pictures

The pictures explaining the addresses belonging to a selected G-function. These pictures are available with "assisted entry" and "dialogue entry" of the control.

b. Dynamical graphic.

With "assisted entry" and "dialogue entry" of the control there is a 2D wire plot graphic available for showing the programmed contour and the cutter path of the tool moving around the contour.

As soon as a few blocks are entered the dynamical graphics can be used to show the effect of the programmed blocks. So changes can be made immediately.

The window defined in a G195 block is used in this graphic mode.

c. Interactive Contour Programming (ICP)

The dynamical graphic within ICP allows to see on the display each program block. ICP does not only show the blocks which are ready, but also indicates the possible choices, such as the intersection point of a line and a circle or two circles.

If elements cannot be drawn, ICP shows in a special window up to three elements which are programmed, but cannot be used up to now. Once the control can calculate the end points of the blocks, all elements are shown on the graphic.

Depending on the ICP-contour size, a special window is used in the ICP mode for the geometry section of the program.

14.4 Machine constants

General

MC 11: define the plane to be set at start up of the control
 =17: XY-plane; =18: XZ-plane; =19: YZ-plane.

MC 19: Type of projection with 2.5D graphic modes
 =0 european projection
 =1 american projection

MC 20: Axes configuration with 2D and 2.5D graphic modes.
 G17 X-axis horizontal and Y-axis vertical
 G18 Z-axis horizontal and X-axis vertical
 G19 Y-axis horizontal and Z-axis vertical
 =0 zero point lower left
 =1 zero point upper left
 =3 zero point upper right
 =4 zero point lower right

MC 78: M13 and M14 are recognized by the control.

MC 82: Maximum number of defined points which can be stored in the control.

MC 83: Maximum number of E-parameters which can be stored in the control.

MC 85: Maximum number of partprograms and macros which can be stored in the control.

MC 135: Feed limitation threshold value

MC 136: The corner release distance

MC 137: The maximum corner error

Programming

MC 707: =70 control starts up in inch mode
 =71 control starts up in metric mode

MC 711: Angle for intermediate circle

MC 714: Type of scaling parameter and the axes involved.
 =0 scaling with a factor; main axes only
 =1 scaling with a percentage; main axes only
 =2 scaling with a factor; main axes and tool axis
 =3 scaling with a percentage; main axes and tool axis

MC 715: Format of the scaling factor

MC 720: Overlap parameter with pocket milling (cutting width)

MC 723: Deceleration distance with G84

MC 724: Dwell at the bottom with tapping G84

- MC 727: Minimum spindle speed with deceleration with G84
- MC 728: Pitch reduction with tapping G84
- MC 740: Maximum feed rate

Measuring with a touch trigger probe

- MC 840: Measuring probe connected
- MC 841: Type of measuring probe
 =0 an inductive probe
 =1 a remote signalling probe (eg. an infrared probe)
 =4 a hard wired probe
- MC 842: Duration of air blow before measuring
- MC 843: Measuring feed rate
- MC 844: Pre-measuring distance
- MC 845: Post-measuring distance
- MC 846: Display resolution for the rotary axis with axis alignment
- MC 847: Width fixed measuring probe
- MC 848: Radius calibration ring
- MC 850: Collision protection measuring probe
 =0 during all movements except the measuring movement
 =1 during all feed movements except the measuring movement
 =2 during all movements except the measuring movement and the retract after the measurement.
- MC 851: Delay time for the collision protection

Tool handling

- MC 27: Number of tools in the tool memory
- MC 28: Number of places in the tool magazine
- MC 29: Tool life monitoring activated
- MC 31: Cutting force monitoring activated
- MC 32: Tool breakage monitoring activated

External program selection

- MC 42: =0 no external program call
 =1 a fixed assignment of the program name
 =2 a variable assignment of the program name

MC 43: With external program call the number of variable assignments which can be stored in the control.

MC 44: =0 G14 and G29 are executed as described in the manual. The execution is not influenced externally.
 =1 G14 and G29 are executed as described in the manual but they can be influenced externally.

Communication

MC 771: =0 no check on block numbers
 =1 check on same block numbers

MC 772: =0 no syntax check on the entered block
 =1 a validation check on the entered block

MC 773: =0 Blocknumbers < 9000
 =1 Blocknumbers >= 9000

14.5 External program selection

When automatic workpiece changers (eg. pallet stations) are used, it is necessary to activate and to execute a new partprogram without interference of the user.

The external partprogram selection allows:

- to assign to each partprogram a special identification;
- to store the identification of the next program during the execution of a partprogram;
- to activate the requested partprogram after finishing the running program (at M30);
- to execute the program after an external start command.

Two possibilities are available for the external selection of a partprogram:

1. FIXED ASSIGNMENT.

The last three digits of the identification number of the partprogram can be entered into the control via the interface of the machine tool.

In the partprogram the relation between the mounting zero point C and the machine zero point Mi can be established with one of the functions for the zero offsets (G54 - G59, G54I[nr.]) and the preset values (G52).

2. VARIABLE ASSIGNMENT.

A special three-digit identification number is assigned to a partprogram. These three digits can be entered into the control via the interface of the machine tool.

There is a special memory in the control, the external program call memory, in which are stored:

- the special identification number (E-word). The E-word ranges from 0 to 999, so 1000 programs can be assigned. The E-words are stored in the memory in increasing order. A machine constant determines how many assignments can be stored in this memory;
- the program identification (N-word) of the partprogram which is activated when the E-word is selected;
- the pallet offsets belonging to the pallet used with the workpiece.

PALLET OFFSETS WITH VARIABLE ASSIGNMENT



NB8613

The pallet offsets are related to the secondary machine zero point M_i . These offsets define the mounting zero point C .

After the external activation of a program the pallet offsets are loaded into the zero offset memory under G52 and this function is automatically activated.

Because G52 is automatically activated, it is advisable to end each partprogram with G51. This method prevents the pallet offsets of a previous program to be used in the program that follows.

Note:

Two machine constants must be set:

- one for activating the external program call function and to indicate what type of assignment is used.
- one for stating how many assignments can be stored in the memory of the control.

INDEX

ABSOLUTE PROGRAMMING	397	EMERGENCY STOP	539
ACCELERATION FACTOR	119	END POINT WINDOW	388
ACCELERATION REDUCTION	4, 119, 440	ETHERNET LINK	502
AIR BLOW	150, 693	F-FUNCTION	17, 25, 37, 54
ARITHMETIC FUNCTION		FMS TOOL MEMORY	343, 347, 534
abs	543, 545, 547, 548, 551	G0 RAPID TRAVERSE	21
arccos	3, 543, 547, 548	G1 LINEAR INTERPOLATION	24
arsin	3, 543, 547, 548	G11	
atan	3, 333, 334, 543, 547, 548, 549	One point geometry	86, 87, 89
ceil	3	Two line geometry	86, 88, 93
cosinus	67, 543, 547, 548	Two point-geometry	86, 87, 91
floor	3, 543, 545	G11 ONE/TWO POINT OR TWO LINE GEOMETRY WITH	
int 543, 544, 545, 546, 548		CHAMFER OR ROUNDING	81
Integer conversion	543	G14 REPEAT FUNCTION	95
maximum	543, 546	G141 3D-TOOL CORRECTION	309
minimum	543, 546	Dynamic TCPM	312
pi 543, 546, 547		Nominal tool	310
Radians (rad)	546, 547	Surface normal vector	311, 312, 316
Reciprocals	548	TCPM	3, 309, 310, 311, 312, 314, 315, 316, 317, 319
sign3, 6, 37, 136, 216, 218, 225, 229, 230, 233, 234, 247,		Tool Centre Point Management	3, 311
251, 252, 255, 259, 260, 263, 267, 270, 273, 274, 279,		Tool vector	312, 316, 317, 319
280, 285, 286, 394, 395, 396, 423, 425, 427, 428, 462,		G145 LINEAIR MEASURING MOVEMENT	321
464, 465, 467, 468, 486, 489, 495, 543, 544, 545, 546		G148 READ PROBE STATUS	340
sinus	543, 546, 547, 548	G149 READ TOOL DATA OR ZERO OFFSET	342
sqrt	338, 543, 545, 548, 549, 551	G150 CHANGE TOOL DATA OR ZERO OFFSET	346
tangens	543, 547, 548	G17 MAINPLANE	98, 556
AXES CHARACTERS	64	G174 TOOL WITHDRAWAL MOVEMENT	349
AXIS CHANGE	685	G18 MAINPLANE	101
AXIS CONFIGURATION	99, 102, 105, 356	G180 BASIC COORDINATE SYSTEM	351
AXIS SELECTION	684	G182 CYLINDRICAL COORDINATE SYSTEM	353
BASIC COORDINATE SYSTEM	351, 368	G19 MAINPLANE	104
BASIC MEASUREMENT MOVEMENT	164, 169	G195 GRAPHIC WINDOW DEFINITION	362
BTR	7, 60	G196 END CONTOUR DESCRIPTION	364
CAD	59, 113, 310, 311, 312, 316, 317, 319, 686	G197/G198 BEGIN INNER/OUTER CONTOUR	
CALIBRATION MANDREL	471	DESCRIPTION	365
CALIBRATION RING	161, 162, 521, 693	G199 BEGIN CONTOUR DESCRIPTION	373
CALIBRATION TOOL	471, 472, 482, 483	G2/G3	
CARTESIAN COORDINATES... 23, 25, 29, 40, 41, 42, 54, 75,		CCW	18, 35, 242, 244, 436, 511, 517
217, 218, 247, 292, 352, 357, 366, 375, 410, 415		Circular arc	193, 225, 233, 241, 252
CLAMPING STATION	11, 183, 187	Circular interpolation	18, 686
CLIMB MILLING	274, 281, 286	clockwise	35, 244
CONFIGURATION FILE	459, 462, 467	Counter clockwise ... 48, 225, 282, 287, 391, 425, 511, 517	
CONSTANT CUTTING FEED (F1=) ... 138, 139, 437, 503, 504		CW	18, 35, 242, 244, 436, 511, 517, 556
CONTOUR ACCURACY	119	G2/G3 Circular interpolation	35
CONTROL PANEL	5, 7, 187, 188	G22 MACRO-CALL	107
CONVENTIONAL MILLING	274, 281, 286	G23 MAIN PROGRAM CALL	113
COOLANT	472, 474, 476	G25/G26 ENABLE/DISABLE FEED- AND/OR SPEED-	
COORDINATE SYSTEM... 8, 11, 26, 140, 144, 184, 187, 188,		OVERRIDE	115
345, 348, 352, 353, 357, 358, 360, 361, 377, 679, 680		G25/G26 FEED- SPEED OVERRIDE 115, 125, 440, 553, 686	
COORDINATES CROSS	65, 685	G27/G28 POSITIONING FUNCTIONS	116
CORNER ACCURACY	60	G29 CONDITIONAL JUMP	120
CUTTING FORCE MONITOR	533, 538, 693	G300 PROGRAMMING ERROR MESSAGES	432
CYCLE		G301 ERROR IN A PROGRAM THAT JUST HAS READ IN	
G77 Bolt hole circle	241	 434	
G79 Activate cycle	250	G302 OVERWRITING RADIUS COMPENSATION	
G81 Drilling cycle	254	PARAMETERS	435, 560
G83 Deep hole drilling cycle	257	G303 M19 WITH PROGRAMMABLE DIRECTION	436
G84 Tapping cycle	262	G319 QUERY ACTUAL TECHNOLOGY DATA	437
G85 Reaming cycle	266	G320 QUERY ACTUAL G DATA	439
G86 Boring cycle	269	G321 QUERY TOOL DATA	443
G88 Groove milling cycle	278	G322 QUERY MACHINE CONSTANT MEMORY	445
G89 Circular pocket milling cycle	284	G324 QUERY G-GROUP	446
CYCLE DESIGN	3, 4, 466, 493, 501	G325 QUERY M-GROUP	448
DATA COMMUNICATIONS PROGRAM	502	G326 QUERY ACTUAL POSITION	450
DECELERATION DISTANCE	692	G327 QUERY OPERATION MODE	452
DEFINING COORDINATES	14	G329 QUERY OFFSET FROM KINEMATIC MODEL	453
DISPLAY RESOLUTION	693	G331 WRITE TOOL DATA	454
EDIT MODE	387	G339 WRITE OFFSET IN KINEMATIC MODEL	457
EDITING	8, 386	G350 DISPLAY WINDOW	463

G351 WRITE TO FILE	466
G36/G37 ACTIVATE/DEACTIVATE TURNING MODE	123, 553, 555
G39 ACTIVATE/DEACTIVATE TOOL OFFSET	124
G4 DWELL TIME	51
G40 CANCEL TOOL RADIUS COMPENSATION	128
G41/G42 TOOL RADIUS COMPENSATION	132
G43/G44 TOOL RADIUS COMPENSATION TO/PAST ENDPOINT	143
G45 AXIS PARALLEL MEASURING MOVEMENT AND MEASURING TOOL DIMENSIONS	147
G46 MEASURING A FULL CIRCLE OR PROBE CALIBRATION	156
G49 CHECKING ON TOLERANCES	164
G50 PROCESSING MEASURING RESULTS	169
G51/G52 CANCEL/ACTIVATE PALLET ZERO POINT SHIFT	19, 181, 183, 296, 297
G53/G54 CANCEL/ACTIVATE ZERO POINT SHIFT	19, 182
G6 SPLINE-INTERPOLATION	52
G61 TANGENTIAL APPROACH	191
G62 TANGENTIAL EXIT	196
G63/G64 GEOMETRIC CALCULATIONS	20, 201, 410, 412, 416, 424
G66/G67 DETECT TOOL DIRECTION	23, 105, 224, 225, 252
G7 TILTING WORKING PLANE	61
G70/G71 INCH/METRIC PROGRAMMING	18, 227, 503
G72/G73 SCALING OR MIRROR IMAGING	20, 229, 296
G74 ABSOLUTE POSITION	237
G78 POINT DEFINITION	246
G8 TILTING TOOL ORIENTATION	69
Compensation movement	72, 73, 315, 316, 686
G9 DEFINE POLE POSITION	74
G90/G91 ABSOLUTE/INCREMENTAL PROGRAMMING	18, 57, 74, 289, 290, 292, 313, 359, 410, 412, 416
G92/G93 INCREMENTAL/ABSOLUTE ZERO POINT SHIFT	11, 19, 40, 41, 60, 184, 188, 231, 232, 233, 234, 246, 294, 295, 297, 315, 345, 352, 355, 357, 402, 406, 441, 472, 683
G94/G95 SELECT FEEDRATE UNIT	19, 223, 302, 359, 503, 557
G96/G97 CONSTANT CUTTING SPEED	304, 558
G98 GRAPHIC WINDOW DEFINITION	305
G99 DEFINITION OF WORKPIECE BLANK AS A BOX	307
GENERAL PROGRAMMING INFORMATION	5
GEOMETRIC CALCULATIONS	18, 223, 597, 598, 643
Circle and line	598, 638, 653
Concentric circles	598, 642
Connecting circle	210, 598, 635, 636, 638, 640, 641, 643, 650, 652, 654, 656, 658, 669, 671, 672, 673, 674
Connecting circle indicator	598, 643
Intersection point	136, 203, 205, 206, 207, 219, 598, 599, 601, 602, 622, 624
Intersection point indicator	205, 219
Line and circle	598, 636, 652
Parallel line	218, 663
Tangency indicator	209
Two circles	598, 640
Two lines	217, 550, 598, 599, 602, 649
HELIX	27, 36, 192, 197, 223, 402, 406
HELIX INTERPOLATION	223, 402, 406
HIGH-PERFORMANCE ALGORITHMS	686
IDENTIFICATION NUMBER	8, 325, 513, 526, 528, 530, 531, 532, 535, 695
IMPROVEMENTS	3
INCH	18, 227
INFRARED PROBE	522, 693
INPOSITION	117
INTRODUCTION	1, 17, 381, 493, 553, 679, 686
JOG OPERATION	64, 683
KINEMATIC MODEL	3, 72, 315, 349, 350, 453, 457, 458
LASER	4, 222, 323, 341, 469, 470, 471, 472, 475, 477, 480, 481
G600 Calibration	470
G601 Measure tool length	473
G602 Measure length and radius	475
G603 Check of individual edge	477
G604 Tool breakage control	479
LOOK AHEAD FEED (LAF)	116, 118, 686, 687
MACHINE BUILDER HANDBOOK	449, 539
MACHINE CONSTANT	18, 26, 158, 163, 356, 388, 398, 445, 469, 481, 483, 485, 486, 487, 489, 491, 496, 524, 554, 692, 693, 694
MACHINE REFERENCE POINT	9, 10
MAGNETIC DIGITAL CASSETTE	5
MAIN AXIS COORDINATES	229
MANUALS	1
MEASURING	20, 147, 148, 151, 152, 154, 155, 156, 157, 160, 161, 321, 325, 326, 327, 338, 339, 402, 406, 458, 469, 481, 520, 521, 522, 686, 693
MEASURING DISTANCE	149, 150, 158, 159, 163, 693
M-FUNCTION	18, 51, 150, 223, 264, 310, 325, 330, 448, 449, 524
MIRRORING	234, 235, 236, 402, 406, 441
NESTING	385, 387, 388, 548
NESTING LEVEL	385, 387
OFFSET	
Tool length	124, 125, 440
Tool radius	124, 440
OFFSET PROGRAMMING	125
OPERATING MANUAL	10, 155
OPERATOR MACHINE CONSTANTS	469
OVERRIDE	
Feed	19, 115
Speed	115
PALLET	181
PITCH OF HELIX	42, 43
PLANE FEED	504
PLANE TILT G7	63
POLAR COORDINATES	15, 41, 97, 292, 366, 368, 375
POSITION AXIS	684
POSITIONING LOGIC	118, 119, 440
PRINT FILE	459, 462
PROBE CALIBRATION	20, 156, 161
PROGRAM BLOCK	7, 97
PROGRAM WORD	6, 17
REFERENCE POINT	684
REMOTE SIGNALLING PROBE	522, 693
RETAP THREAD	264
SCALING	60, 231, 441
SCALING FACTOR	232, 692
S-FUNCTION	18, 529
SOFTWARE LIMIT SWITCH	10, 350, 363
SPACE VECTOR	680
SPARE TOOL	343, 347, 531, 532, 535, 537, 538
SPEED SUPERIMPOSE SWITCH	476, 478
SPINDLE SPEED RANGE	524, 529
SPLINE	
Bezier-Splines	52, 53, 54, 56, 57
Cubic splines	53
Radius of curvature	58
Spline	20, 52, 53, 58
SQUARE-HEAD PROBE	153, 154
STANDARD CONFIGURATION	356
S-WORD	536
SYNCHRON GRAPHICS	60, 306, 363, 688
T-FUNCTION	18, 530
THREAD MILL CUTTER	50
TOOL CHANGE	67, 68, 402, 406, 530
TOOL CORRECTION	321, 322, 559, 561
TOOL DIMENSIONS	99, 102, 105, 170, 559
TOOL ORIENTATION	311, 444, 455, 534, 568, 570, 572, 574, 576, 578, 580, 582, 584, 586, 588, 590, 592
TT130	
G606 Calibration	482
G607 Measuring tool length	484
G608 Measuring tool radius	488
G609 Measuring tool length and radius	490
Stylus	161, 469, 471, 481
TURNING CYCLE	
G691 Unbalance measurement	563
G692 Unbalance check	566

G822 Clearance axial	569	G203 Start pocket contour description	409
G823 Clearance radial	571	G204 End pocket contour description	412
G826 Clearance axial finishing	573	G205 Start island contour description	414
G827 Clearance radial finishing	575	G206 End pocket contour description	419
G832 Roughing axial	577	G207 Call island contour macro	422
G833 Roughing radial	579	G208 Quadrangle contour description	425
G836 Roughing axial finishing	581	Islands 383, 407, 408, 416, 417, 418, 420, 421, 423, 424,	
G837 Roughing radial finishing	583	429, 430	
G842 Grooving axial	585	Pseudo-Island	382
G843 Grooving radial	587	VERSION V410	1, 4, 194, 198, 469
G846 Grooving axial finishing	589	VERSION V420	1, 3, 313, 496, 497, 498, 543, 545
G847 Grooving finishing	591	VERSION V500	1, 3
TURNING MODE	3, 123, 304, 372, 435, 553, 554	VERSION V501	447
UNBALANCE		WINDOW SIZE	238, 240
ARRnnnnn.CFG File	459, 460, 461	WIREPLOT GRAPHIC	367, 376
Tabel number	461	WORDWISE	22, 25, 37, 289, 292
Tabel read function	459, 461	WORKPIECE ZERO POINT	74, 75, 78, 182, 186, 423, 553
Unbalance	3, 554, 562, 564, 567	ZERO POINT SHIFT	
UNIVERSAL POCKET		Extended MC84>0	169, 182, 186, 342, 346
Finishing macro	386, 389	Standard MC84=0	169, 182, 342, 346, 434
G201 Start contour pocket cycle	394		
G202 End contour pocket cycle	406		